AWTEC 2012
Asian Wave and Tidal Conference Series
Nov. 27-30, Jeju Island, Korea

CFD study of a ducted cross flow turbine concept for high efficiency tidal current energy extraction

Incheol Kim[1], Joji Wata[1], M. Rafiuddin Ahmed[2], Youngho Lee[3,#]

1 Graduate School, Dept. of Mechanical Engineering, Korea Maritime University, Busan, 606-791, Korea
2.Division of Mechanical Engineering, The University of the South Pacific, Suva, Fiji
3. Division of Mechanical and Energy Systems Engineering, Korea Maritime University, Busan, 606-791, Korea.
Corresponding Author / E-mail: lyh@hhu.ac.kr, TEL: +82-51-410-4293, FAX: +82-51-403-0381

KEYWORDS : Tidal current energy; Cross-flow turbine; Renewable energy.

There is a growing interest in utilizing tidal current energy for power generation which has led to extensive research on this source of renewable energy. The work presented here aims to study the tidal current energy extraction using a cross-flow turbine. The bi-directional flow of tidal currents is used to drive a uni-directional cross-flow turbine. The cross-flow turbine studied is placed in an augmentation channel, having a nozzle and a diffuser. The performance of the device is studied numerically using the commercial ANSYS-CFX code. The internal flow characteristics of the turbine is studied for various cases. Results of the numerical analysis are presented in terms of pressure contours, streaklines, velocity vectors, power coefficient, and performance curves.

NOMENCLATURE

A = initial turbine inlet size, m^2
= tip speed ratio(TSR)
C_p = power coefficient
R = blade radius, m
V_R = tidal current speed, m/s^2
= angular velocity, m/s^2
ρ = density of water, kg/m^3
U= tidal current speed, m/s^2

1. Introduction

There are many forms of ocean energy that can be converted such as tidal range, ocean currents, tidal currents, wave energy etc... From these various types of ocean energy, marine current energy has the advantage of having a predictable steady power output that can be supplied over a long period of time. However, a disadvantage is that the terrain distribution in various areas may affect the water speed adversely and reduce it below the design speed. Since power output is proportionally related to the cubic power of velocity, a lower velocity in a certain area will decrease the feasibility of installation. Therefore, more study is needed for the research and development of low speed tidal turbines in order to increase the performance in low speed areas and as well as improve the efficiency of tidal turbines in suitably chosen sites.[1]

Previously, research was done on Bi-Directional cross flow turbine by Kim et al.[1] It was seen that the augmentation channel strongly influenced the flow and the turbine performance. In addition, the flow resistance by the augmentation channel and the blades forced the flow to divert away from the augmentation channel, reducing the flow velocity and turbine efficiency.

In this paper, a duct was applied onto the previous Bi-directional turbine to study the internal fluid flow of the duct and the turbine, the influence on turbine performance according to the duct size and to find an optimized duct by CFD in order to increase the efficiency of the turbine.[1] In addition, the turbine performance variation with tip speed ratio was investigated in order to obtain the highest efficiency of the optimized duct. [1]

2. Conceptual design of the turbine system

2.1 Tip speed ratio (TSR)

The Tip speed ratio is an important design variable of the rotor blade. The angular velocity can be calculated using equation (1). In this equation, is the tip speed ratio, R is the blade radius, is the angular velocity and V_R is the design current velocity.

$$\lambda = \frac{R\omega}{V_R} = \frac{Tip\ speed\ ratio}{Tidal\ Current\ Speed} \qquad (1)$$

When the tip speed ratios is higher than the appropriate tip speed ratio, vibration and cavitation will occur. On the other hand, at lower tip speed ratios, the output power will be relatively small. The Tip speed range near the highest CP point in the performance curves by Tip speed requires a detailed analysis. The optimum Tip speed range can be defined at about 80-90% on either side of the maximum Cp point but the range can be differed according to the performance curves. [1]

The cross flow turbine was made to achieve the highest CP at a tip speed range near 1.[1]

2.2 Cross flow turbine model

In order to apply cross flow turbines for tidal power generation, it is necessary for the fluid direction and angle to develop the optimum angle of attack on the blade. As the working fluid enters the inlet and if it completely fills the area of the turbine, part of the flow will cause the rotation of the turbine whereas other parts will resist the rotation in the opposite direction. This will reduce the output power of the turbine. Therefore, the turbine is not fully exposed to the flow but the flow must be guided via instead to rotate the turbine in the direction of the blade. In addition, another important variable is the outlet angle of the runner blade. If the outlet flow angle from the inlet is not optimum, the flow may interfere with the rotation of the turbine near the outlet and cause a decrease in power. [1]

To satisfy these necessary conditions, the nozzle was installed into turbine inlet area. Also, to produce maximum torque, the design of the blade and the angle of the nozzle to turbine inlet area was considered.[1]

The blade length is 464 mm, the axial length is 3,200 mm, the number of blade is 30 and the thickness of the blade is 26.24 mm. The geometry is shown in Figure 1. The model of the augmentation channel is presented briefly in Figure 2. The dimensions of the turbine and the entrance region, is shown in table. 1.

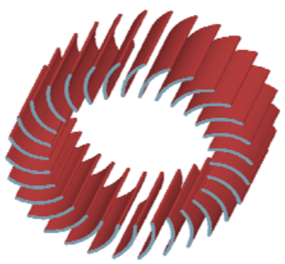


Fig. 1 Rotor blade model

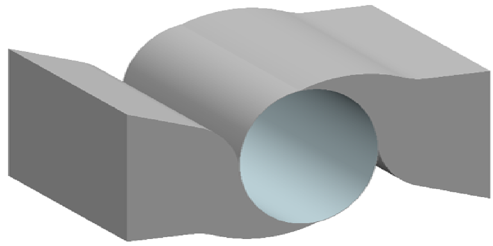


Fig. 2 Modeling of augmentation channel for the turbine

Table. 1 TCP turbine blade design parameters

3-D cad tool	Nx-6
Number of blades	30
Runner inlet angle	30
Runner outlet angle	90
External diameter	1696mm
Internal diameter	768mm
Length of blade	464mm
Thickness of blade	26.24mm
Axial length of blade	3,200mm

2.3 Numerical Analysis

Figure. 3 shows cross flow turbine in the duct. The duct length is 6 m and the initial turbine inlet size(A) is fixed at 3.2 m x 1.6.m . For the initial investigations, the turbine performance characteristics are analyzed by varying the duct inlet area. The duct inlet area sizes are set by increasing the gap distance between the turbine and the duct wall. Figure 4 shows the vertical (Y) and horizontal distances(X) between the turbine and the duct. For each case, the distances Y and X were increased proportionally by a factor. The Y distance was calculated by multiplying the turbine height (C) by the factors 0.25 for the first case,0.5 for the second and by 1 for the third case. In the same manner, the X distance varied by multiplying the width (B) with the same factors. This resulted in the following duct area sizes of 2.25A, 4A and 9A as various cases. The duct inlet boundary condition is set as normal speed with a water velocity is 2.5 m/s. The outlet condition is set as relative pressure at 0 Pa and the reference pressure is at atmospheric pressure. In addition, the walls are modeled under free slip condition..

A general frozen rotor interface connection was used in the portion between the turbine and nozzle.

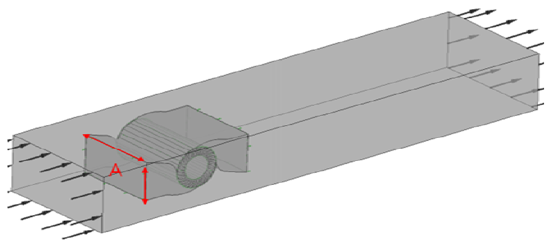


Fig. 3 Computational domain of turbine

The analysis was first calculated for each case of duct size (2.25A, 4A, 9A) at 14RPM. From this simulation the best case was selected to be analyzed further. At this duct size, the tip speed ratio was varied from 0.8-1.4 and the turbine performance characteristics was further analyzed .

The Torque and angular velocity calculated from this simulation was used to calculate the Cp by using equation 4.

$$C_p = \frac{T\omega}{\frac{1}{2}\rho A U^3}$$

In the present analysis, hexahedral grids were generated to obtain higher accuracy. The generated mesh is shown in Figures 5 to 6.The total number of nodes at the blade section is approximately 900,000. The augmentation channel section has approximately 500,000 nodes. The number of nodes in the domain surrounding the turbine is approximately 400,000. The total number of nodes was approximately 1,800,000. To carry out the CFD analysis, 6 parallel computers were used. A commercial CFD code of CFX-13 is adopted to conduct CFD simulation to solve incompressible turbulent flow. To achieve good convergence of the results, the k- is applied as the turbulence model.

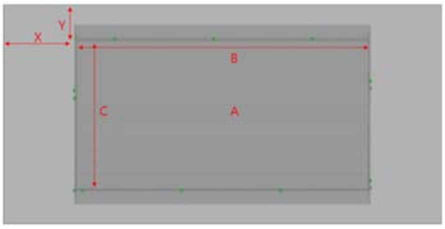


Fig. 4 Nomenclature of the turbine and Duct geometry

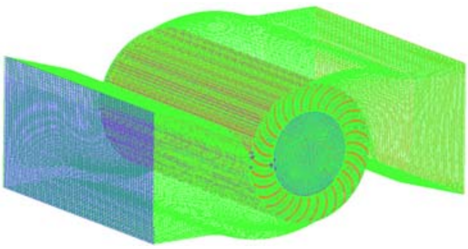
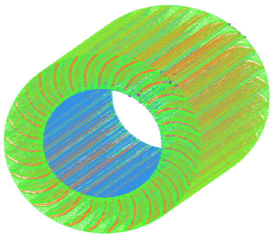


Fig. 5 Grid system of turbine



. Fig. 6 Grid system of runner

2.4 Results and Discussion

Figure 7-13 shows results of the analysis by duct size. The figures 7 to 9 shows velocity vectors from the duct inlet to the turbine and as well as the flow around it. From these figures, the case with the reduced turbine inlet area (2.25A) had the fastest flow velocity within the turbine. Also, the figures show the flow separating from the top side of the turbine due to the sharp corner. Whereas the flow follows the casing at the bottom side of the inlet before separating near the diffuser. The flow around turbine is accelerated through the gap between the duct and turbine. The flow around the diffuser section is seen to interact with the outflow from the diffuser. In the figure 7, the high speed external flow influences the flow from the diffuser and the flow is seen to become stable compared to figures 8 and 9. For future investigations, a modified case with a straight external wall can be used to accelerate the flow around the turbine in order to study the effect on turbine performance.

The Pressure distribution contours at various inlet areas are shown in figures 10-12. These figures show a high pressure region near the inlet with the highest pressure recorded near the edges. As the flow enters the turbine region, the velocity is seen to increase as shown in previous figures. As this occurs the pressure is seen to decrease. In addition, as the duct area increases, the pressure before and inside the turbine inlet is decreased and the cases with the increased duct area have a relatively lower maximum high pressure at the turbine inlet. In the high pressure region, the velocity vectors had shown a lower velocity than the duct inlet velocity of 2.5m/s. As the water passes through the turbine and into the lower pressure region, the velocity is seen to increase and in some areas have a higher velocity than the duct inlet velocity. From these 3 cases, the case with the highest pressure difference and higher velocity vectors is seen in the 2.25A duct size case.

The performance curves of the 3 cases are shown in figure 13. The best case was the 2.25A case with an efficiency of 0.563, a power output of 22.46kW. From this figure, it was seen that as the duct area increases, the performance characteristics of the turbine decreased.

Figure 14 shows power coefficient calculated from the torque of five cases in the tip speed ratio range from 0.8 to 1.4 at the turbine inlet size of 2.25A. The Cp was over 0.5 in tip speed ratio 0.8 to 1.4. The maximum power coefficient 0.563 at a tip speed ratio of 0.99. The rated RPM for this turbine is 14RPM at the design flow speed.

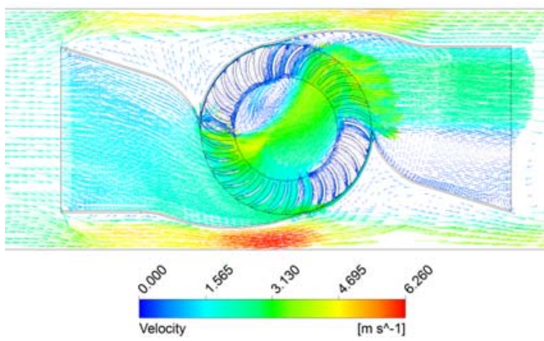


Fig. 7 Velocity vector at 2.25A duct size

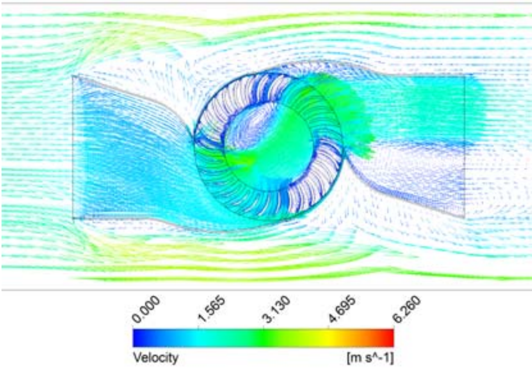


Fig. 8 Velocity vector at 4A duct size

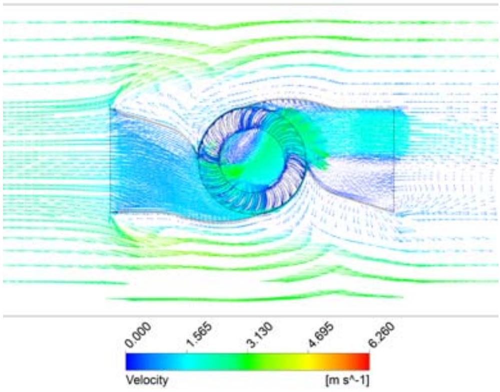


Fig. 9 Velocity vector at 9A duct size

The performance curves are shown in Figure 15. The performance curves, such as torque, are important for power systems and generator design and control . The maximum torque of 17,429 Nm occurs at a tip speed ratio of 0.85 but the maximum output power of 22.46 kW and Torque of 15321 Nm occurs at a tip speed ratio of 0.99

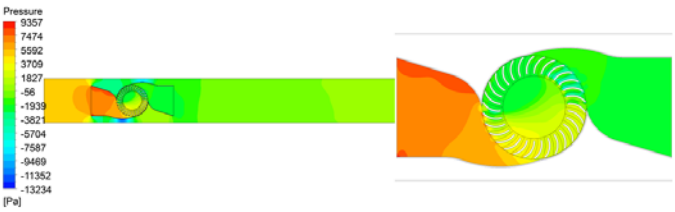


Fig. 10 Pressure contours of the flow area at 2.25A duct size

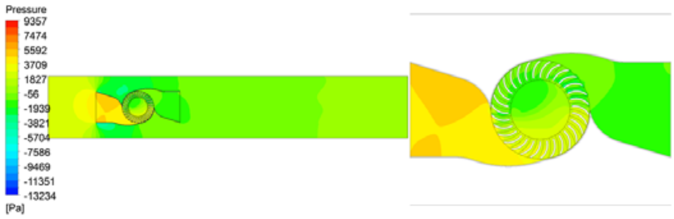


Fig. 11 Pressure contours of the flow area at 4A duct size

Fig. 12 Pressure contours of the flow area at 9A duct size

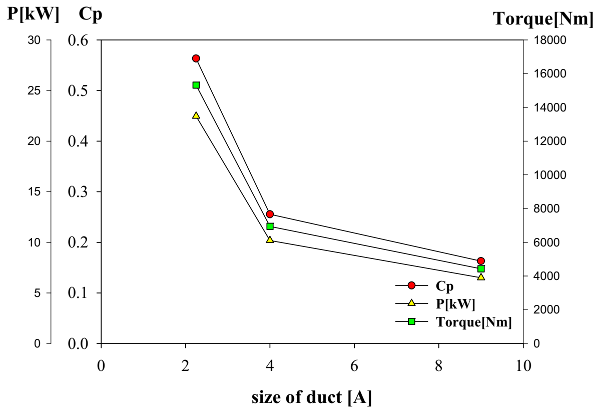


Fig. 13 Performance curves according to size of duct

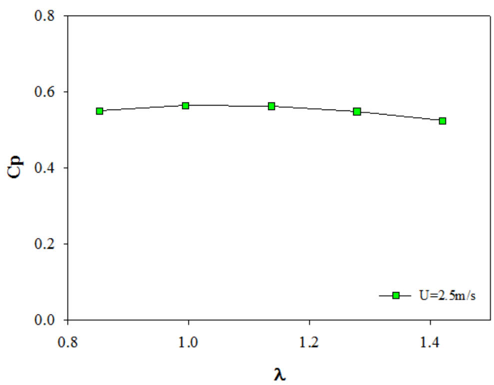


Fig. 14 Cp curve according to TSR

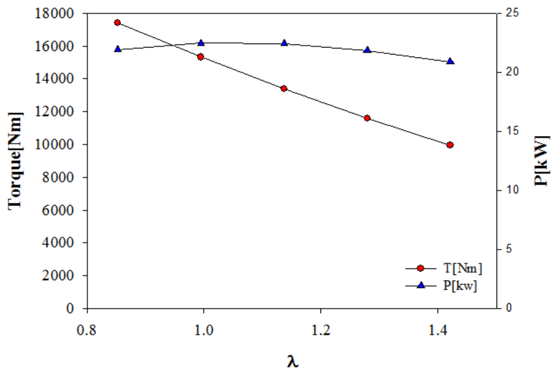


Fig. 15 Performance characteristics curve according to TSR

3. Conclusions

This study focused on the influence on the turbine performance according to the variation of the duct size. The external casing shape and external flow of duct was not considered but was taken as default from the conceptual design stage from previous research. Figure 7 to 13 shows that as the duct size increases, the flow speed is reduced and therefore it reduces the performance characteristics. The case with the highest performance was the 2.25A duct size case. From figures 14 and 15, it was seen the optimum tip speed ratio for the 2.25A duct size case was at TSR of 0.99.In addition, mixing effects between the faster flow at the outside the casing and the outlet flow may have increased flow speed of the outlet flow due to the gap and may have improved performance. Therefore, the gap around the turbine casing may have an significant influence. For future studies, The influence of casing design on the external flow can be investigated.

ACKNOWLEDGEMENT

This work is the outcome of a Manpower Development Program for Marine Energy by the Ministry of Land, Transport and Maritime Affairs(MLTM)

REFERENCES

1. Kim, K., Ahmed, M., Hwang, Y. and Lee, Y., “Conceptual Design of a 100kW Energy Integrated Type Bi-Directional Tidal Current Turbine”, Proc. of the 10th AICFM. 2009, Paper 212.
2. CHOI, Y., LIM,J., KIM, Y., and Lee,Y., "Performance and Internal Flow Characteristics of a Cross-flow Hydro Turbine by the Shape of Nozzle and Runner Blade", J. of Fl. Sci. and Tech. 2008, Vol.3 NO.3, pp.398-409
3. Lim, J., “Blade configuration study to improve the performance of horizontal axis tidal current turbine”, 2011 Inha Uni. master's thesis
4. Kim, K., Ahmed, M., and Lee, Y., "Efficiency Improvement of a tidal current turbine utilizing a larger area of channel", Ren. En., Volume 48, December 2012, Pages 557-564
5. Choi, Y., Kim, C., Kim, Y., Song, J., Lee, Y., "A performance study on a direct drive hydro turbine for wave energy converter." J. of Mech. Sci. and Tech. 2010;24:2197-206.