A Study on the Design and Performance Prediction of MW Class Ocean Current Turbine

Bum-Suk Kim[1], Woo-June Kim[1] and Mann-Eung Kim[2]

[1] New Growth Technical Support Team,
Korean Register of Shipping
23-7 Jang-dong Yuseong-gu Daejeon, Republic of Korea
E-mail: bskim@krs.co.kr

[2] Energy & Environmental Business Center,
Korean Register of Shipping
23-7 Jang-dong Yuseong-gu Daejeon, Republic of Korea
E-mail: mekim@krs.co.kr

Abstract

Ocean energy resources such as sea temperature, wind, wave and current have potential powers which can be converted into electricity, among them a current turbine is worthy of note since it is less sensitive to the environmental circumstances including wind velocity, temperature, and wave condition. Rotor blade is a one of the important device which changes kinetic energy of ocean current to mechanical energy. It affects power performance, load and dynamic stability of turbine system both directly and indirectly. The purpose of this study is to design a baseline blade for 1MW horizontal ocean current turbine using blade element momentum theory(BEMT), and conduct CFD analyses to predict flow field around the blades and performance characteristics such as thrust force, efficiency and power output with changes to tip speed ratio(λ). And also, design reference data such as pressure, streamline, torque and thrust distributions on the blade surface is presented as well to investigate optimal design parameters.

Keywords: Ocean current turbine, Blade design, CFD, Cavitation, Performance prediction

1. Introduction

Seawater temperature, wind, wave, tide and current have potentials which can be used to ocean energy conversion. Among these, ocean current is worthy of note as it is less sensitive to climate changes (wind, wave and others) compared with other energy sources.

Moreover, current change can be predicted precisely and accurate economical effects can be estimated through calculating AEP (Annual Energy Production).

However, there are some weaknesses including problems occurring due to characteristics of ocean current turbine such as waterproof, rust protection, seabed installation and operation and it increases the costs of installation and maintenance compared with wind turbine. This paper will describe the characteristics of KR 19-2C an unique blade which is designed for 1MW horizontal axis ocean current turbine based on BEMT (Blade Element Momentum Theory)[1]. In addition, CFD simulation is performed to estimate blade power coefficient according to changes in TSR (tip speed ratio, λ) and examine flow characteristics and cavitation phenomenon.

2. Blade Design

Ocean current turbine can produce electricity by transforming kinetic energy of ocean current into rotating force of blades. Therefore, it is conceptually the same with wind turbine except its working fluid is seawater instead of air. However, blades used for ocean current turbine could experience serious damages caused by cavitation according to the tip speed of the blade. It can reduce the endurance of ocean current turbine below the design life time, thus careful attention to avoid cavitation must be paid at blade design stage. Blade Element Momentum Theory which is used in wind turbine blade design is also applied in the same way, and design value of λ is set to 6 to generate 1MW power at rated current speed of 2.5 m/s. Detailed variables for blade design are listed in Table 1 and 3D shape of the designed blade is shown in Figure 1.

3. CFD Simulation and Analysis Results

3.1 Computational grids and simulation conditions

This computational study consists of two parts; one is steady state analysis to predict power performance and flow field around the blade and the range of cavitation phenomenon occurred in steady state also

Table 1: Design specifications of KR 19-2C

Design parameters	Values
Prated (rated power)	1 MW
Cp(estimated power coefficient)	0.48
η(drive train efficiency)	0.9
Vrated (rated current velocity)	2.5 m/s
ρ (density)	1,024 kg/m^3
D (diameter)	19.2 m
N (number of blades)	2
ω (rotational speed)	14.92 rpm

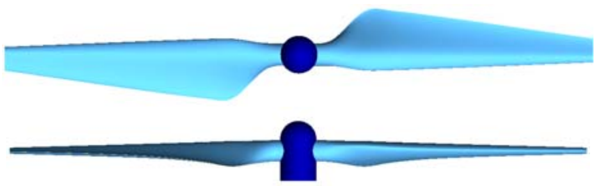

Figure 1: 3D model of the KR 19-2C

can be examined, and the other is transient analysis to examine the changes of cavitation distribution according to time domain at 2D section around the blade tip (μ=0.95). For 2D section transient simulation, a section located at 95% from the 3D blade hub is chosen and the approaching velocity and inflow angle are calculated at the condition of the rated operating condition. Calculation conditions for 2D CFD simulation are listed in Table 2. The calculation domain to predict power coefficient and 3D flow characteristics contains only one blade and periodic condition is applied to each boundary surface. The simulation was performed as λ changes from 2 to 9 and cavitation prediction model is applied. Calculation conditions for 3D CFD simulation are listed in Table 3. ANSYS ICEM-CFD V11.0 is used to make computational grids and both of 2D and 3D simulation model are generated by Multi-block HEXA grid system. About 7.2 million nodes are used in 3D simulation and 2D simulation contains 0.2 million nodes. All computational grids were maintained y+ condition under 5 at the surface of the wall to apply transitional turbulence model. Figure 2 shows computational grids for each simulation case. type.

3.2 Surface pressure & streamlines

Figure 3 shows the calculated result of pressure distribution and streamlines on 3D blade surface at TSR of 3, 6 and 9. At pressure side of the blade, the highest pressure is predicted at the leading edge of a blade around the tip region and the pressure drops dramatically at 25% point from the leading edge. If the pressure level predicted by simulation is below the saturation vapour pressure of seawater, it is highly possible that cavitation will occur. The range of low pressure area tends to be widened and the amount of pressure drop becomes higher as TSR increases. However, the contrary tendency is investigated at the leading edge of the suction side, the size of low pressure area and the amount of pressure drop increases

Table 2: Calculation conditions of 2D foil – S814

Case	1	Density	1024 kg/m^3
AOA	3.5 deg	$P_{saturated}$	17.4 mmHg
Section	95 %	Chord	748.07 mm

Table 3: Calculation conditions of 3D blade

Case	R. speed rps	TSR	Temp. °C	Density kg/m^3	$P_{saturated}$ mmHg
1	1.56	9	25	1024	17.4
2	1.56	8	25	1024	17.4
3	1.56	7	25	1024	17.4
4	1.56	6	25	1024	17.4
5	1.56	5	25	1024	17.4
46	1.56	4	25	1024	17.4
7	1.56	3	25	1024	17.4
8	1.56	2	25	1024	17.4

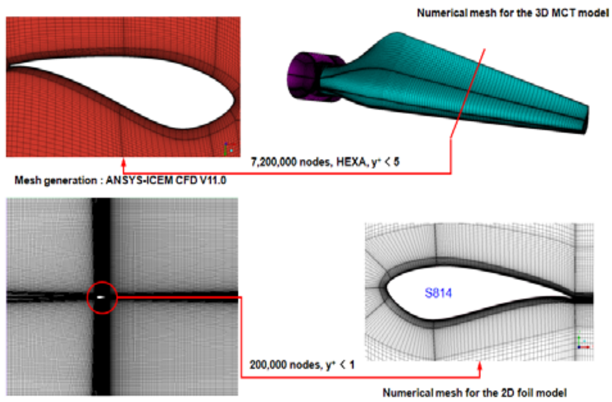

Figure 2: Computational grids

as TSR decreases. Under most conditions, streamlines on the surface of pressure side form relatively uniform attached flow. Good attached flows are also observed at the suction side except for an area around hub when TSR has the value of 6 or 9. At TSR is 3, however, attached flow is rarely formed and fairly complicated streamlines are observed on the surface of suction side.

This indicates that a stall at the suction side is deeper and deeper as current velocity increases and it can be the major reason for declining of ocean current turbine's efficiency.

3.3 Cavitation

Unlike wind turbine, the ocean current turbine is installed and operated under the water and the blades which are installed in the shallow water might experience cavitation problem according to the changes of blade azimuth angle. Cavitation could cause erosion of blade surface or vibration problem which in turn will shorten the design life of whole system and cause frequent maintenance and repair. Therefore, investi-gation on the cavitation phenomena considering installation depth of ocean current turbine should be done when designing a new blade for ocean current turbine. Figure 4 shows the experimental result preformed by Bahaj et al[2] in 2007 and the generation of sheet and cloud cavitation at blade tip is clearly identified. Since the rated tip speed of the designed blade for 1MW power generation reaches 15m/s, sheet and cloud cavitation might be appeared around the tip region where pressure drops dramatically. To simulate

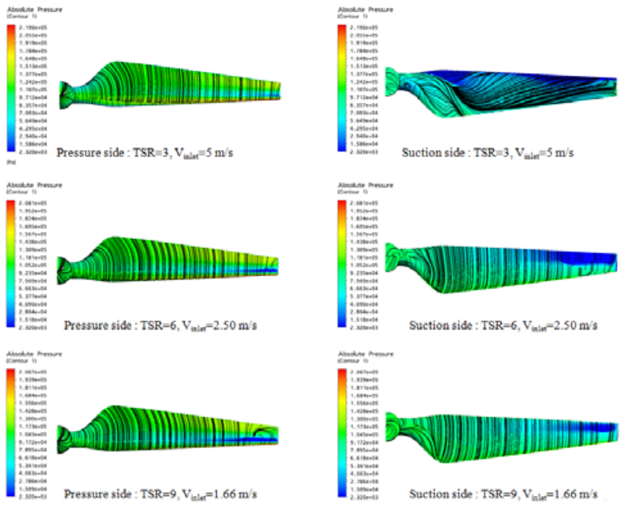

Figure 3: Surface streamlines & pressure distributions

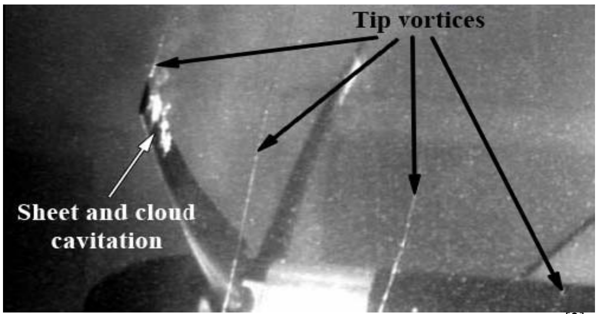

Figure 4: Cavitation phenomena on a blade surface[2]

cavitation phenomena accurately, a complete blade model should be included in free surface flow analysis but a full system simulation requires huge amount of computing time and resources, thus the effects of the changes of pressure due to the depth of water is not considered and simulation is performed under the assumption that only a single blade is surrounded by constant hydraulic pressure condition. Figure 5 shows the predicted cavitation region on the designed blades. As mentioned in Section 3.2., cavitation observed around tip of the blades on pressure side when TSR is high and cavitation region at the suction side is expanded as TSR decreases. Especially, in the case of TSR has the value of 3, cavitation occurs at the most of the leading edge area of suction side. To minimize the problems due to cavitation on ocean current turbine, design factors such as selection of hydrofoils that forms the shape of blade, rotational speed and the depth of installation should be considered with great importance at the blade design stage. Figure 6 shows the predicted result of cavitation on the 2D section which is located on non-dimensional distance (r/R=0.95) from the hub to observe the characteristics of cavitation on tip of the blade according to timeline. The region painted to red indicates that the volume of fraction for air is 1 and the region painted to blue indicates that volume of fraction for sea water is 1. As shown in Figure 6, sheet and cloud cavitation occurs repeatedly on suction and pressure sides as time goes by and erosion and vibration problems are expected to occur around the tip of the blade.

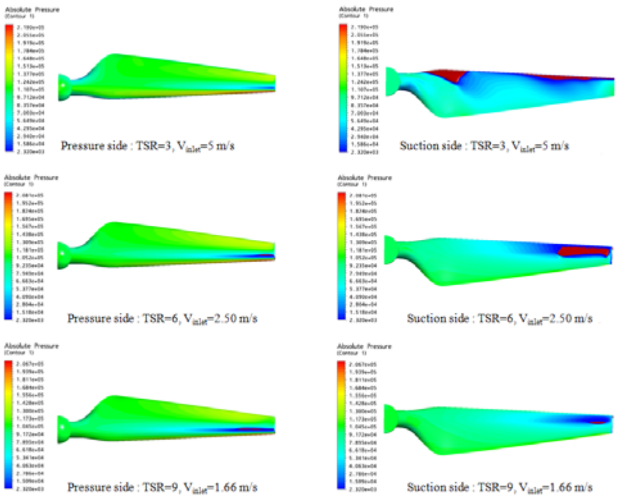

Figure 5: Cavitation phenomena

3.4 Performance prediction

Figure 7 shows expected performance coefficients for designed blade. Maximum performance coefficient is predicted to 0.47 at TSR has the value of 7. As TSR increases, performance coefficient increases until TSR reaches 7 and then it decreases afterward. Since performance coefficients shown in Figure 7 are obtained from aerodynamic power, they always have higher values than performance coefficients based on electrical power which includes the efficiency of drive train and generator. It is well known that the theoretical maximum efficiency of blade under general external flow condition is 59.3% and the aerodynamic efficiency for commercial ocean current turbine is around 48~50%.

Induced drag force increases due to the effect of vortex generated at the blade tip, and it decreases the efficiency of the blade. Thus, the efficiency of blade can be improved if the blade tip has the optimized shape to diminish the tip vortex and it is expected that aerodynamic efficiency of the baseline blade could go up to 48% by optimizing the tip shape.

4. Conclusion

In this study, a blade for 1MW ocean current turbine is designed successfully based on BEMT and CFD simulations considering cavitation effects are performed to predict aerodynamic performance. According to the performance prediction, the maximum efficiency of the newly designed blade is 47% at TSR=7. Blades for ocean current turbine can experience cavitation near the tip area according to its installation depth and rotational speed, so careful attention must be paid at the design stage to avoid problems caused by cavitation.

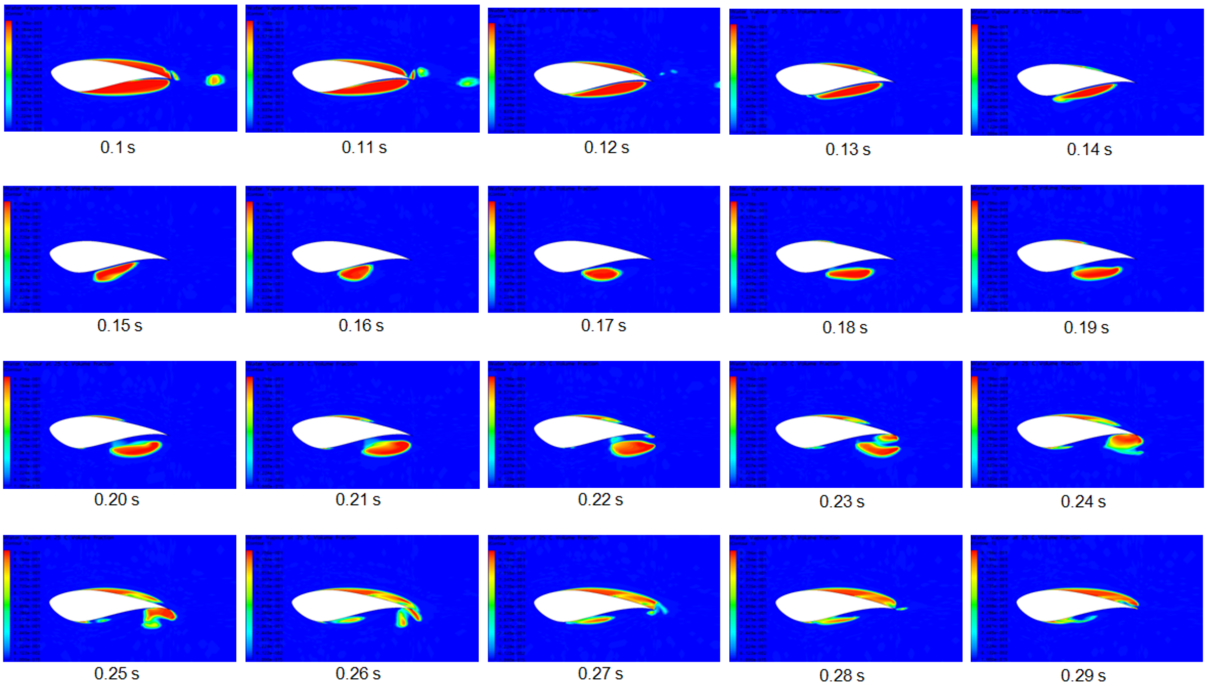

Figure 6: Time series cavitation phenomena (r/R=0.95)

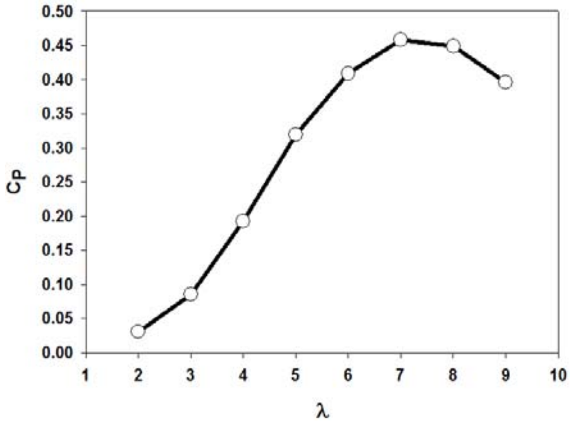

Figure 7: Predicted power coefficient

References

[1] B.S. Kim., (2005): A Study on the Optimum Blade Design and the Aerodynamic Performance Analysis for the Horizontal Axis wind Turbine. Doctoral dissertation, Korea Maritime University.

[2] Bahaj, A.S., Mo;;anf, A.F., Chaplin, J.R., Batten, W.M.J., (2007): Power and thrust measurements of current turbines under various hydro dynamic flow conditions in a cavitation tunnel and a towing tank, Journal of Renewable Energy, Vol. 32 No. 3, pp.407-426.

[3] J.A. Clarke, G. Connor, A.D. Grant, C.M. Johnstone, (2006): Design and testing of a contra-rotating tidal current turbine, Journal of Power and Energy, Vol. 221 Part A.

[4] W.M.J. Batten, A.S.Bahaj, A.G. Molland, J.R. Chaplin, (2008): The Prediction of the hydrodynamic performance of marine current turbines, Journal of Renewable Energy, Vol. 33, pp. 1085-1096.