Numerical simulation of cavitation for a horizontal axis marine current turbine

Q Guo¹, L J Zhou¹,³ and Z W Wang²
¹ College of Water Resources and Civil Engineering, China Agricultural University, Beijing 100083, China
² Department of Thermal Engineering, Tsinghua University, Beijing 100084, China

E-mail: zlj09@263.net

Abstract. Marine current turbines, compared with the analogous wind turbines, have the potential to suffer cavitation. This paper focuses on the effect of cavitation on a marine current turbine and implements the two-phase flow simulations based on the Rayleigh-Plesset cavitation model. It can be found that under the influence of cavitation, the power and thrust coefficients of turbine decrease and especially near the blade tips. Due to the non-uniform hydrostatic pressure along the depth of water, the present work takes this effect into account and finds a larger cavitation area appears on the blade tips at a shallow submergence. Meanwhile the power and thrust performance of one blade change during its rotating period. In order to analyze the cavitation performance of turbine locates at a certain water depth, the variations of inflow velocity and rotor rotating speed are implemented. It indicates that, with the inflow velocity increasing, the $C_P$ and $C_T$ of turbine with higher rotation speed will significantly reduce due to the occurrence of cavitation, and the influence on the turbine with lower speed is small. These predicting results can provide implications for the safe and stable operation of marine current turbines.

1. Introduction
Cavitation is a general fluid mechanics phenomenon and usually has a potentially undesirable effect on the hydraulic machinery. Marine current turbines show many similarities in design and operating characteristics to the wind turbines with high-aspect ratio blades, but an important difference is the marine current turbines has the potential to suffer cavitation [1]. Cavitation may damage the blade surface material and affect the operations of marine current turbines.

An investigation of cavitation performance on two-dimensional sections can be used as a starting point in the preliminary design stage for marine current turbines, as the cavitation tunnel experiments and the numerical simulations for several suitable sections have demonstrated that how the section shape and some parameters of the foil affect the cavitation performance [1-2]. The three-dimensional cavitation flow conditions has been shown by a test in a cavitation tunnel with a model of the marine current turbine, and the investigation has indicated that careful design of blade angles and tip speeds should be possible to prevent cavitation on full scale marine current turbines [3]. Wang D et al [4] found the turbine can experience strong unstable sheet and cloud cavitation as well as strong tip vortex

³ Corresponding author.
cavitation at a shallow depth by experimental results. Batten W M J et al. [5] predicted that the cavitation could occur on both sides of the blade and both for the design case or when the blade is heavily pitched for power regulation, so all areas of operation would need to be considered when designing a rotor. According to the definition of the cavitation number, which is a function of pressure and velocity, it can be inferred that the likelihood of cavitation is related to the locations upon blades, specifically the greatest chance of cavitation damage will occur at the blade tips when at the highest point of rotation [6]. Due to the variation of the hydrostatic pressure in different water depth, the cavitation number in one point on the blade will change with the rotor rotating, especially on the blade tip of a large turbine [7]. Based on the interest of the cavitation characteristics in the process of rotating, this paper simulates a marine current turbine model with the effect of the vertical hydrostatic pressure. The tests of cavitation are always carried out in a fully enclosed tunnel by reducing the pressure, where the pressure field differs from the undersea environment for the prototype turbine, thus the present work studies the effect of incident flow velocity on the cavitation.

2. Methodology

2.1. Computational domain and mesh generation
The model turbine was from the cavitation tunnel tests of Bahaj A S et al [3] with a 0.8 m diameter \(D\) rotor arranged to the fully enclosed tunnel with the size of 5 m \(\times\) 2.4 m \(\times\) 1.2 m. The experimental results with 20° hub pitch angle and \(U_T = 1.4\) m/s at zero yaw were used as the reference for the simulations. The computational domain included two parts as the rotating inner domain with the rotor located at the coordinate system’s origin and the stationary outer domain with symmetrical sizes in both height and width directions. The unstructured tetrahedral mesh was generated in the inner domain and the structured hexahedral mesh was for the outer domain. Mesh sensitivity was guaranteed by comparison with the variation of mesh nodes number. Figure 1 shows the schematic computational domain and the mesh structure in a cross section.

2.2. Numerical simulation
A constant inflow condition of \(U_T\) was applied to the inlet with 2\(D\) upstream distance from rotor. Opening condition with different pressures based on the cavitation numbers \(\sigma\) defined as equation (1) [3] was set in downstream.

\[
\sigma = \frac{P_T - P_V}{0.5 \rho V^2}
\]  

(1)

where \(P_T\) is the tunnel pressure, \(P_V\) is vapour pressure with the value of 2000 N/m\(^2\), \(\rho\) is the fluid density and \(V\) is referred to the velocity at the blade tip as \(V = \sqrt{U_T^2 + (\omega R)^2}\) relating to the angular velocity \(\omega\) and rotor radius \(R\).

Numerical simulations were performed in ANSYS CFX with RNG \(k-e\) turbulence model and Rayleigh-Plesset cavitation model. The high resolution scheme was used for the advection scheme.
3. Results and discussion

3.1. Cavitation features and effects

The simulation type is transient with the consideration the effect of gravity. According to the experimental results that cavitation appeared until the cavitation number was reduced to about 0.9 and there was strong cavitation with cavitation number was below about 0.4, so the experimental condition of $\sigma=0.44$ at TSR=8.7 was compared with a non-cavitation condition with $\sigma=1.34$ ($P_r=101325\ N/m^2$) at the same TSR. For marine current turbines, the non-dimensional power and thrust coefficients ($C_p$ and $C_T$) were defined as $C_p=T\omega/(0.5\rho U_r^3A)$ and $C_T=F/(0.5\rho U_r^2A)$, where $T$ and $F$ is the torque and thrust, and $A$ is rotor swept area. Considering the blockage corrections [3], figure 2 shows the azimuthal variation of corrected $C_p$ and $C_T$ for the single blade (dash line) and for the whole rotor (solid line) on both $\sigma=0.44$ and $\sigma=1.34$ conditions. The azimuthal angle $\theta$ was defined as the locations of the coloured blade in its period of rotating clockwise shown in figure 2.

![Figure 2. Azimuthal variation of $C_p$ and $C_T$ for the single blade and rotor.](image)

Compared with the available data from the towing tank experiment with 20° hub pitch at 1.4m/s towed speed in [3], the deviation for simulation results of $\sigma=1.34$ were less than 5% for $C_p$ and 1% for $C_T$. This deviation is mainly due to the differences between the test tank and cavitation tunnel results compared in the experiment and the simplified rotor geometry without support structures used in the simulation. The calculations of $\sigma=1.34$ can be used as a validated reference for the cavitation condition ($\sigma=0.44$) within the error range. Figure 2 reveals the power performance was negatively affected by the cavitation with the declined $C_p$ values, but the effect on the thrust performance was small. It is interesting that, for the condition of $\sigma=0.44$, the single blade has the maximum value for $C_p$ with its tip near the lowest point of rotation, and the $C_p$ values decline with the higher location of the single blade. The variation of $C_T$ values has the same trend within small scope.

In order to further analyse the effect from cavitation on the power and thrust performance for the single blade, figure 3 shows the radial variation of $C_p$ and $C_T$ for three azimuthal locations on both non-cavitation and cavitation conditions. Here the single blade except the blade root is divided into 16 radial slices whose $\delta C_p$ and $\delta C_T$ can be obtained.

It can be seen that the effect on $C_p$ and $C_T$ from cavitation mainly appears near the blade tip region within $r/R>0.7$ and it is more serious when the blade locates at the higher point of rotation with $\theta=0$. Considering the hydrostatic pressure varied with the water depth, it can be speculated that the pressure is lower with a shallow depth of submergence and where the cavitation will be more serious. According to the relationship between the negative pressure coefficient ($-C_{\text{pressure}}$) and the cavitation number ($\sigma$) [8], figure 4(a) shows the distribution of $-C_{\text{pressure}}$ for a foil near blade tip (at $r/R=0.9$) at three azimuthal locations. It indicates that the cavitation mainly appears near the trailing edge on the suction side of blades, and the radial length of cavities on blade ① is longer than that on the other
blades slightly shown in figure 4(b) by the iso-surface of the vapour volume fraction. On the pressure side, there is even small region with cavitation near the leading edge for the blade with shallow depth of submergence, shown in figure 4(a, c).

![Figure 3](image)

**Figure 3.** Radial variation of $\delta C_P$ and $\delta C_T$ for the single blade at different azimuthal angle.

![Figure 4](image)

**Figure 4.** Cavitation position and the distribution of pressure coefficient on a foil section.

Based on the variation of cavitation region and the power and thrust performance mentioned above, one blade operates in the rotating period will undergo the alternating effect and may lead to the fatigue damage or noise. The main factor is the hydrostatic pressure in different water depth and its effect will be more serious with the size of turbines increasing [7]. Experimental study on cavitation always use the closed tunnel with pressure decreased, but the ambient pressure is relative higher when the prototype turbines operate in river or ocean. In addition to the effect of pressure, the cavitation number is related to the velocity as shown in equation (1), thus the next section focus on the effect of velocity on the cavitation.

### 3.2. The effect of velocity on cavitation

The marine current turbines always locate at a relative deep depth of water to avoid interference of waves [9], and then the ambient pressure will be higher than an atmosphere generally. In order to simplify the calculation model, the free surface is neglected and the ambient pressure is set as an atmosphere which represents a shallow submergence for the turbine. The velocity $V$ in equation (1) can be transformed into $V=U_T \sqrt{1+\text{TSR}^2}$ which is related to the inflow velocity $U_T$ and the non-dimensional TSR. The mean peak tidal currents are generally over 2m/s in the world [5]. Figure 5 shows the power and thrust performance with blockage corrections of present turbine under different TSR conditions with the various inflow velocities. At the same time, the cavitation numbers ($\sigma$) at...
each condition are shown near the marks. The cavitation regions are shown with the distribution of pressure in figure 6 on some typical conditions. Due to the limit time, these results are from the steady simulations.

![Figure 5: Power and thrust performance of the turbine under different inflow velocity.](image)

There is no distinct cavitation in the simulations with $U_T=1.5$ m/s and the most suitable condition is near TSR=6 which was demonstrated in the experiment [3]. Thus the performance can be used as non-cavitation results in the subsequent analysis. For the calculations with $U_T=2.5$ m/s, the $C_P$ has a decreasing trend at higher TSR with a lower value of $\sigma$, as shown in Figure 6(c) with cavities near blade tip, while the $C_T$ still agrees well with the non-cavitation result. With the increasing of $U_T$, the optimal condition with maximum $C_P$ value moves to the lower TSR due to the occurrence of serious cavitation at higher TSR conditions where the $C_T$ decreases significantly. It is worth noting that Figure 6(d) shows the cavitation appears on the leading edge in a long and narrow region, but its power and thrust performance has almost not been affected. Until $U_T$ increases to 3.5 m/s, there are also some reduction on $C_P$ and $C_T$. In general, the cavitation comes from the variation of inflow velocity has more serious effect on the conditions at higher TSR. Therefore, properly reducing the rotational speed of rotor may be beneficial to prevent the cavitation. However for a specific power output, the rotor operates at lower speed need to support a larger torque [7], which requires a stronger structure. In figure 5 and 6 the model turbine is located at a shallow submergence with an atmosphere as the ambient pressure approximately. With the submerged depth increasing, the ambient pressure will rise and lead to a high cavitation number, thus the cavitation performance will be improved with the cavitation area reducing. According to the distribution of pressure on the suction side in figure 6, it can be speculated that the variation trend for the cavitation region. Considering the blockage effect in the confined tunnel, the cavitation characteristics cannot be corrected thus a further research in a wider computational domain is needed.

4. Conclusions
The two-phase flow calculations using CFX’s homogenous two-phase model together with the Rayleigh-Plesset cavitation model was performed on a marine current turbine model. Some observations made as results are as follows.

(1) Based on the unsteady simulation including the hydrostatic pressure varied with the water depth, it validates that the more serious cavitation occurs on the blade tips when at the shallow water depth of submergence. The turbine shows a significantly reduction on $C_P$ but very little change on $C_T$, and these effects from cavitation are embodied in the region near blade tips especially. Meanwhile, the power and thrust performance of one blade will change but the performance of the whole rotor is almost invariable in its rotating period.

(2) Assuming the turbine locates at a certain water depth, the variation of inflow velocity has effects on the cavitation characteristics. With the incident velocity increasing, the $C_P$ and $C_T$ of turbine with higher rotation speed will significantly reduce due to the occurrence of cavitation. In contrast, there is
small influence on the turbine operating at lower TSR conditions although some cavities appearing on the leading edge of blade.

\[ U_T = \begin{align*}
2.5\text{m/s} & : (a) \sigma = 1.88 \\
3.0\text{m/s} & : (d) \sigma = 1.3 \\
3.5\text{m/s} & : (g) \sigma = 0.96 \\
\end{align*} \]

\[ \text{TSR}=4 \quad \text{TSR}=6 \quad \text{TSR}=8 \]

\[ U_T = \begin{align*}
2.5\text{m/s} & : (b) \sigma = 0.86 \\
3.0\text{m/s} & : (e) \sigma = 0.6 \\
3.5\text{m/s} & : (h) \sigma = 0.44 \\
\end{align*} \]

\[ \text{TSR}=8 \]

\[ U_T = \begin{align*}
2.5\text{m/s} & : (c) \sigma = 0.49 \\
3.0\text{m/s} & : (f) \sigma = 0.34 \\
3.5\text{m/s} & : (i) \sigma = 0.25 \\
\end{align*} \]

**Figure 6.** Cavitation positions and pressure contours on the suction side under different inflow velocity.

**Acknowledgements**

The authors thank the Open Research Fund Program of State Key Laboratory of Hydrosience and Engineering (No. sklhse-2012-E-01) and the Specialized Research Fund for the Doctoral Program of Higher Education (No. 20130002110072) for supporting the present work.

**References**

[1] Batten W M J, Bahaj A S, Molland A F and Chaplin J R 2006 *Renew. Energy* **31** 249-56
[2] Molland A F, Bahaj A S, Chaplin J R and Batten W M J 2004 Measurements and prediction of forces, pressures and cavitation on 2-D sections suitable for marine current turbines *Proc. Inst. Mech. Eng. Part M: J. Eng. Marit. Environ.* **218** (2) 127-38
[3] Bahaj A S, Molland A F, Chaplin J R and Batten W M J 2007 *Renew. Energy* **32** 407-426
[4] Wang D, Atlar M and Sampson R 2007 *Proc. Inst. Mech. Eng. Part A: J. Power Eng.* **221** 219-31
[5] Batten W M J, Bahaj A S, Molland A F and Chaplin J R 2008 *Renew. Energy* **33** 1085-96
[6] Bahaj A S and Myers L E 2001 Cavitation prediction in operating marine current turbines *Proc. Conference C76 of the Solar Energy Society* (Belfast, UK, 6-14 Sept 2001)
[7] Ying Y, Li W, Liu H W and Lin Y G 2008 *Compressor Blower & Fan Technology* **4** 8-11
[8] Zhao W G, Zhang L X, Xin X P, Shao X M and Li W 2011 *Jixie Gongcheng Xuebao* **47** 171-6
[9] Bahaj A S, Myers L E, Rawlinson-Smith R I and Thomson M 2011 *J. Offshore Mech. Arct. Eng.* **134** 021104