Flow analysis of NACA0015 and its application on tidal power

D M Liu1, 2, H Chen1, L L Yuan1, Y Z Zhao1 and X B Liu2

1 Research & Development Center of Dongfang Electric Machinery CO., LTD, Deyang, 618000, Sichuan Province, China.
2 Fluid machinery key laboratory of Xihua University, Chengdu 610039, China

E-mail: liudemin999@126.com

Abstract. NACA0015 is a commonly used hydrofoil for hydraulic machinery. Because the calculated flow characteristic is sensitive to the mesh density and distribution, a careful check of mesh independence should be performed before the CFD calculation. For this purpose, flow comparison was done between calculation and experimental test at the same operating condition with the Reynolds number equal to 130,000. When the span length is two times of the chord length, the ratio of lift to drag is 34.97, which is close to the test result of 36.47. The pressure coefficient and cavitation characteristic were also calculated. For example, the pressure coefficient was compared in different flow conditions with Reynolds number equal to 130,000, 200,000 and 400,000. The cavitation characteristic was calculated at different cavitation numbers, e.g. 0.5, 1.0 and 1.5. Based on these hydraulic and cavitation characteristic, a tidal turbine runner was developed by using hydrofoil NACA0015. According to our studies, the flow characteristic and cavitation index can satisfy the engineering requirements.

1. Introduction
In China and many other countries there is an emerging market for green electricity derived from renewable energy resources. Among the green energy resources is oceanic energy: for instance there are waves, currents and ocean thermal energy. To humankind, it is a serious challenging to meet the future’s energy demands without harmfully impacting on the environment through getting power either by consuming more fossil fuels or relying more on nuclear energy. In this situation, developing oceanic energy resources might be a solution.

Recent studies [1, 2] indicate that marine currents have the potential to supply a significant fraction of future electricity needs. Therefore many countries have invested to develop technologies in this field. Various types of tidal turbines have been reported such as horizontal axial, vertical axial and open-center turbines. Whatever type of rotor is used, the theoretical upper limit of efficiency for extracting energy from a free stream is 59.3 percent, which can be derived from actuator disk theory—a conclusion commonly attributed to Betz [3-4]. In practice, a tidal turbine rotor, which generally will be large in relation to the flow cross-section (i.e. with its upper and lower edges close to the surface) may benefit from an effect called ‘blockage’. This is where the flow is constrained by the boundaries so that it cannot diverge to the extent that it would when decelerated through a rotor in free space, and as a result a greater mass flow of fluid passes through the rotor and this manifests itself as a higher efficiency [5-6].
In this paper, a bulb-typed tidal turbine, whose blades were shaped based on NACA0015 hydrofoil, was proposed and tested by numerical simulations. This design is simple compared to other horizontal turbines because of the complex seal structure for protecting the generator. This paper first focused on the flow analysis of NACA0015 hydrofoil in respect with hydrodynamic characteristics, and introduced some details of the mathematical model and mesh schemes used in the analysis. It then described the simulation results of hydrofoil under full wet and cavitation condition. At the end, it concluded that NACA0015 could be a better solution for tidal turbine design.

2. Mathematical model

In homogeneous multiphase model, a common flow field is shared by all fluids, as well as other relevant fields such as turbulence. Thus, the governing equations of homogeneous model for mass, momentum and volume conservation can be written as

\[
\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m \vec{u})}{\partial x_i} = 0
\]

(1)

\[
\frac{\partial (\rho_m \vec{u})}{\partial t} + \frac{\partial (\rho_m \vec{u} \cdot \vec{u})}{\partial x_j} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \bar{F}_v - \rho_m \bar{u} \bar{u}_j \right)
\]

(2)

\[
\frac{\partial (\rho \alpha_j)}{\partial t} + \frac{\partial (\rho \alpha_j \vec{u}_j)}{\partial x_j} = \rho_j \dot{m}
\]

(3)

where

\[
\bar{F}_v = \begin{cases} 
\mu_m \left( \frac{\partial \vec{u}_i}{\partial x_j} + \frac{\partial \vec{u}_j}{\partial x_i} \right) & i \neq j \\
-\rho_m \bar{u} \bar{u}_j = \mu \left( \frac{\partial \vec{u}_i}{\partial x_j} + \frac{\partial \vec{u}_j}{\partial x_i} \right) - \frac{2}{3} \rho_m k \delta & i = j
\end{cases}
\]

\[
\mu_m = \alpha \mu_v + (1 - \alpha) \mu_t
\]

All kinds of cavitation models [7-8] which is provided by many researchers are applied on cavitation phenomena. Such as Singhal (2002) [7] and Zwart-Gerber-Belamri (2004) [8] give different cavitation equations and cavitation coefficient. The source terms [9] \( \dot{m} \) can be expressed as

If \( p < p_v^* \):

\[
\dot{m} = C_1 \frac{3(1 - \alpha_v - \alpha_u)}{r} \frac{2\sigma}{2 - \sigma} \left( \frac{M}{2\pi R} \right)^{1/2} \left( p_v^* - p \right)
\]

(4)

If \( p > p_v^* \):

\[
\dot{m} = C_2 \frac{3\alpha_v}{r} \frac{2\sigma}{2 - \sigma} \left( \frac{M}{2\pi R} \right)^{1/2} \left( p - p_v^* \right)
\]

(5)

\[
p_v^* = p_v + \frac{P_{turb}}{2} = p_v + 0.195 \rho k
\]

(6)

3. Calculation domain and Boundary condition of hydrofoil

Figure 1 is the calculation domain of hydrofoil, the length of hydrofoil is \( C=0.1m \) and the inflow angle is 9\(^\circ\). To avoid the inflow condition and wall affection, the distance 5.3\( C \) and \( C \) from inflow and wall is set. The detail boundary conditions are shown in Table 1.
Table 1. Boundary condition

| Parameter                        | Value         | Reference       | Condition       |
|----------------------------------|---------------|-----------------|-----------------|
| The chord length of the hydrofoil| 0.1m          | Hydrofoil wall  | wall condition  |
| The span length of the hydrofoil | 0.08m         | Upper plane     | Wall condition  |
| Incidence angle                  | 9°            | Lower plane     | Wall condition  |
| Fluid material                   | water         | Cavititation number | 0.5, 1.0, 1.5 |
| Inlet velocity                   | 1.3m/s, 2m/s, 4m/s | Turbulence model | RNG $k-\varepsilon$ |
| out                              | pressure      | Simulation type | steady          |

The structure of hydrofoil mesh is finished on the X-Y plane as shown in Figure 1. When the node number is 400,000, the calculated ratio between lift and drag is 34.97, which is close to the test result of 36.47 as shown in Figure 2. To be insensitive to the calculation results, the mesh with the node number of 800,000 was selected in this paper.

Figure 1. Model and flow conditions

Figure 2. Lift and drag coefficient and node number

4. Calculated results and discussion of hydrofoil

The pressure coefficient $C_p$ was compared between test and calculation as shown in Figure 3 and the consistency is excellent.

Figure 3. Pressure coefficient between test and calculation

Figure 4. Pressure coefficient at different inflow conditions
Furthermore, the pressure coefficient was compared under different Reynolds number conditions of 130,000, 200,000 and 400,000. The pressure coefficient distributions are nearly identical in this inflow velocity range, as shown in Figure 4. From low velocity to high velocity, the hydrodynamics of the NACA0015 foil is excellent in wet condition.

![Figure 5](image)

**Figure 5.** Cavitation bubble shape at different cavitation numbers (The red curve on hydrofoil means the volume of cavitation bubble): (a) $\sigma = 0.5$; (b) $\sigma = 1.0$; (c) $\sigma = 1.5$.

The cavitation bubble shape was calculated at different cavitation numbers $\sigma (-C_p)$, e.g. 0.5, 1.0 and 1.5 (shown as in Figure 5). The cavitation bubble shape is sensitive to the cavitation number. From Figure 5 to Figure 6, we can see the volume (red circle) and length of cavitation bubbles increase with the decrease of cavitation number.

![Figure 6](image)

**Figure 6.** Pressure coefficient with cavitation numbers (The horizontal section in the curve means the length of cavitation bubble)

5. **Calculated results and discussion of Tidal turbine**

Based on the above research results, a new tidal turbine runner is developed by just applying the hydrofoil NACA0015. It has 10 blades, please refers to Figure 7. The calculation domain is a big box to simulate the flow domain in the ocean, please refers to Figure 7 (a). The tidal turbine domain mainly includes inlet section, rotor section and outlet section, please refers to Figure 7 (b). The rotor section is the rotation part whose rotation speed is 12r/min. The inlet section is the contraction cone, which can increase the inflow velocity to collect more energy. The outlet section is expanding cone which can recover energy. The detail boundary conditions are shown in Table 2. The blade setting angle $\beta$ is the included angle between blade center line and rotor axis in Figure 7 (c).

The velocity was imposed on the inflow1 of the calculation domain as the inlet condition for the numerical simulation. And the interface boundary condition was imposed on the inflow2 of the tidal turbine as the inlet condition for the numerical simulation.
The output power is calculated at different install angle $\beta$ (Beta) in Figure 8. From the Figure 8, the output power 13KW is maximum value when the setting angle is 24°. The flow discharge is gradually decreasing with setting angle increasing in Figure 9. Because the upwind flow area increases with the blade setting angle increasing.

| Table 2. Boundary condition |
|-----------------------------|
| Fluid material | water |
| Inlet flow 1 velocity | 2m/s--4m/s |
| 1.5 | Rotor stator |
| Moving condition | Calculation domain and turbine |
| Turbulence model | Simulation type |
| RNG $k - \varepsilon$ | steady |

The output power is an important parameter to determine the performance of the tidal turbine, which is calculated by Eq. (7). The power was calculated at different flow velocities of 2.0m/s, 2.5m/s, 3.0m/s, 3.5m/s and 4.0m/s. Figure 10 described the power output of the turbine in relation with the inflow velocity, taking into account the influence of the cavitation. From it we can see that the maximum output of 13KW is achieved with inflow velocity equals 4.0m/s and free of cavitation. When cavitation takes place, the outputs of the turbine runner are clearly reduced at all selected operating points; however, the maximum output power keeps still at 12KW.
6. Conclusions

In this paper, the flow characteristics of the hydrofoil NACA0015 and a bulb-typed tidal turbine were analyzed and discussed, and got the following conclusions.

1) Simulation calculations and tests have been done to NACA0015 hydrofoil at different cavitation numbers and Reynolds numbers. The hydrodynamic characteristic and free cavitation of NACA0015 hydrofoil is adapted for the flow velocity range.

2) The tidal turbine developed by applying the hydrofoil NACA0015 is reliable and the developed tidal turbine output power is 13KW. The maximum output power appears at the blade setting angle 24°. However, the output power will further improve in the next research.

Nomenclature

$C_1$-evaporation coefficient 0.13
$C_2$-condensation coefficient, 0.01
$C_p$-Pressure coefficient
$M$-torque
$N$-rotational speed(r/min)
$P$-output power (MW)
Re-Reynolds number
$p$-pressure (Pa)
$p_v$-vapor pressure (Pa)
$r =$0.0001
$u$-Velocity
$\alpha_v$-Water vapor void fraction

$\alpha_{\infty}$-non-dissolved gas void fraction, which takes value of 0.0005.
$\sigma$ = cavitation number ($-C_p$)
$p_m$-mixture density
$\rho$-water vapor density
$\rho_1$-water density
$\mu$-mixture viscosity
$\mu_v$-water vapor viscosity
$\mu_1$-water viscosity

$k$: Turbulent kinetic. $i, j = 1, 2, 3$.

Acknowledgments

This work is supported by the Chinese National Foundation of Natural Science (No. 51279172), the Open Research Funded Project of Key Laboratory of Fluid Machinery in Sichuan Province (Xihua University) (Grant No.Szjj2012-041, szjj2011-039).

References

[1] Binnie 1993. Tidal stream energy review: UK DTI. Report ETSU, T=05=00155=REP,
Engineering and Power Development Consultants Limited and McAlpine and Sons Limited and IT Power Limited, UK Department of Energy, UK.

[2] EU Commission 1996 The exploitation of tidal marine currents Report EUR16683EN 23-31

[3] Feasibility study of tidal current power generation for coastal waters: Orkney and Shetland. Final Report, EU contract XVII=4 1040=92-41, ICIT, March 1995.

[4] Cave P R and Evans E M 1986 Tidal stream energy systems: economics, technology and costs Third International Symposium on Wave, Tidal, OTEC and Small Scale Hydro-Energy (Brighton, England, 14-16 May 1986)

[5] HSE UK 1995 Offshore Installations: Guidance on Design, Construction and Certification (London: UK Department of Energy, HMSO)

[6] Kihoh S and Shiono M 1994 Power generation by tide and tidal current. In Proceedings of Advanced Materials '93, Vol. V=A, Eco-materials, pp. 41–44

[7] Singhal A K, Athavale M M, Li H and Jiang Y 2002 J Fluid Eng. 124(3) 617-24

[8] Zwart P J, Gerber A G and Belamri T 2004 A Two-Phase Flow Model for Predicting Cavitation Dynamics Fifth International Conference on Multiphase Flow (Yokohama, Japan, 30 May–4 June 2004)

[9] Liu D M, Liu S H, Wu Y L and Xu H 2009 Numerical Analysis of Airfoil NACA0015 and Centrifugal Pump's Cavitation Characteristic Based On the Thermodynamic Effects Proceedings of 2009 ASME Fluids engineering conference (Vail, Colorado, USA, 2-5 August 2009)