Hydraulic characteristics analysis of a bulb tubular turbine based on the CFD simulation and model test

To cite this article: X Z Zhou et al 2018 IOP Conf. Ser.: Earth Environ. Sci. 163 012046

View the article online for updates and enhancements.

Related content
- Development and optimization design of pit turbine with super low-head C X Yang, Y Zheng, D Qzhou et al.
- The simulation analysis and operation conditions optimization of low-head water bulb tubular turbine based on different guide vane outlet angle Z G Li, B Wang, B Ma et al.
- Comparison of Different Measurement Techniques and a CFD Simulation in Complex Terrain Christoph Schütz, Martin Hofsäß, Jan Anger et al.
Hydraulic characteristics analysis of a bulb tubular turbine based on the CFD simulation and model test

X Z Zhou¹, S H Ahn¹, Z W Wang¹*, Z Y Mao¹, Y Y Luo¹ and Q Guo²

¹ Thermal Engineering, Tsinghua University, 100084, Beijing, China; ² College of water resource and civil engineering, China Agricultural University, 100084, Beijing, China;

wzw@mail.tsinghua.edu.cn

Abstract. With the strong demands on the further development of economy and energy, tidal energy, with own green and renewable advantages, has gained more and more attention, of which as the main application type, bulb tubular turbines have played an important role in the power exploitation. Characteristic of low head and large discharge, flow properties of a bulb turbine turn out to be more complicated, which seriously affects the actual operation performance of the station. To promote wide development of tidal energy, it is ritual to deeply dig into the performance of bulb turbine. In this paper, a bulb-type model turbine used for tidal energy is taken for example. By means of CFD method and model test, hydraulic specifications, like efficiency and cavitation properties, have been obtained and compared in case of different conditions. Through the contrast between numerical simulation and model test, a novel corrected method relative to the low-head tidal turbine is brought out for further adaptation to execute the hydraulic, dynamic and stability analysis.

1. Introduction

With the development of the modern society, problems of the resources lack, ecology protect and new energy utilization have been paid more and more attention for most countries all around the world. Ocean energy, known as “Blue Energy”, can not only alleviate the urgent energy shortage problem once largely explored on the earth, but also be considered as the preferred choice for seeking the clean energy and reduce the CO₂ emission [1]. Tidal energy, as an important part of ocean energy, seems to be the most promising resource to be explored both recently and in the future, and moreover attached with renewable, sustainable and green advantages it has been introduced to be the mainly essential element of the national energy developing strategy, which also provides new commercial opportunities for emerging strategic marine industries [2]. Therefore, it is a matter of cardinal significance to evaluate project sites, design hydropower units and predict unit performances accurately reliably for tidal hydropower stations.

Different from other types of power station, tidal power plants are equipped with tubular turbine that is with the properties of low head and large discharge, and consequently it is much more difficult to accurately estimate and predict the performance of prototype turbines especially for the tidal power plants. Recently, various researchers have focused on developing and predicting the ocean energy conversions [3-6] and obtained positive achievements, for example, the gravity force governs the flow field in low-head units, which induces vertical pressure gradients in the turbine runner, and influences the cavitation performance especially for axial-type hydraulic turbines [7-9]. In addition, under low head
operating conditions transient changes like net head variations, local pressure fluctuations and cavitation in tide levels would be also an influence parameter, which indicates that non-uniform influx causes irregular load on the turbine runner and increases vibration by additional radial forces [10]. Therefore, in order to accurately predict the prototype performances, the numerical method should be improved with a new approach for tidal power units.

This paper is to establish a suitable numerical method for predicting tidal turbine performances, and some problems relative to hydraulic performances are discussed by using the model test and the numerical simulation that can contribute to further optimizing operation of the tidal power units especially for low-head turbine.

2. Numerical method

2.1. Governing equation

The moving fluid complies with two basic rules, that is the homogeneous continuity and the momentum equations, which are listed as following:

\[ \nabla (\rho \mathbf{v}) = 0 \]  \hspace{1cm} (1)

\[ \frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} = -\frac{1}{\rho} \nabla p + \mathbf{f} + \nu \nabla^2 \mathbf{v} \]  \hspace{1cm} (2)

where \( \mathbf{f} \) is the body force, \( \rho \) the fluid density, \( \nu \) the viscosity, \( p \) the pressure and \( \mathbf{v} \) the velocity of the fluid.

2.2. Turbulence model

Spalart proposed a curvature correction on eddy viscosity turbulence models for the system rotation and streamline curvature. By use of adapting the curvature correction model to the shear-stress transport model (SST), the proposed model (SST-CC) was found to be competitive with the Reynolds stress transport model for the accuracy and reliability, which was concluded that the SST model with the curvature correction obtained reliable simulation results. In this paper, the SST-CC turbulence model is used for the simulation.

2.3. Calculation objective

For the further analysis, a kind of model turbine unite from the tidal power plant was chosen, and the model unit, as shown in the figure 1, consists of four main parts, that is so-called spiral casing (SP), guide vane (GV), runner (RV) and draft tube (DT), and the relative parameters are demonstrated in the table 1.

![Figure 1. Sketch of a turbine unit](image)
Table 1. Parameters of the turbine unit

| Unit Type            | Dm (mm) | GV number (/) | RV number (/) |
|----------------------|---------|---------------|---------------|
| One-way operation    | 350     | 16            | 3             |

3. Case study

3.1. Model test

As for case simulation to be further checked, the model test designed for the unit was carried out and the layout of the test rig was shown in the figure 2. The measuring pressure at the turbine inlet and outlet was used for the head calculation $H_M$ and the discharge $Q_M$ as well as the rotating speed $n_M$ can be obtained by use of the installed sensors.

Figure 2. Layout of the model turbine unit test

In order to make test results scientific and reasonable, the test procedure and the data processing abides by IEC code 60193-1999 standard [11] and relative parameters such as $Q_{ED}$, $n_{ED}$, $E_M$ and $\eta_M^*$, which were picked out for comparison, are acquired according to the following equations:

$$E_u = \frac{\Delta \rho}{\rho} + \frac{(v_m^2 - v_{out}^2)}{2}$$  \hspace{1cm} (3)

$$n_{ED} = \left( \frac{n_M g D_w}{E_M} \right)^{0.5}$$  \hspace{1cm} (4)

$$Q_{ED} = Q_u / \left( D_M^2 g E_M^{0.5} \right)$$  \hspace{1cm} (5)

$$\eta_M^* = \eta_u + (\Delta \eta_h)_{M \rightarrow M^*}$$  \hspace{1cm} (6)

where $Q_{ED}$, $n_{ED}$, $E_M$, $\eta_M$ and $\eta_M^*$ are the discharge factor, speed factor, specific hydraulic energy, and turbine efficiency at different and constant Reynolds number $Re$ respectively.

Based on test data, through the transform by the formula, hydraulic specifications of the model turbine in case of different conditions were achieved in the form of the points and change curves, which can be seen in the table 2 and figure 3.
Table 2. Test points

| No. | $n_{ED}$ | $Q_{ED}$ | $\eta_{M}^*$ |
|-----|----------|----------|--------------|
| 1(opt.) | 1.0295 | 0.636 | 93.01 |
| 2 | 1.2108 | 1.04 | 90.68 |
| 3 | 1.2108 | 1.20 | 87.35 |

Figure 3. Test curve of the model unit

Figure 4. Discretization structure of the model turbine

3.2. Numerical simulation

3.2.1. Discretization model and boundary conditions. With the use of CFD method, it is essential to carry out the discretization of the computation domains and in view of the flow passage property and further analysis in detail, the individual calculation models for the whole analysis are shown in the figure 4 and to deal with the interpolation of the different parts, corresponding mesh is refined and interfaces are introduced between each flow part.
Given the individual parts of the flow passage, the calculation model has been obtained by assembling spiral casing, guide vane, runner and draft tube with the help of the interfaces. Based on the connection type, General Connection interface is adopted for static conditions while Frozen rotor interface takes into effect in case of rotor-stator components. Meanwhile the pressure conditions at the inlet and outlet and rotating speed are set corresponding to the objective test points as well.

Since the calculation model is established, it is to determine the mesh independency by evaluating the mesh convergence using various numbers of elements for the whole flow passageway. Table 3 and figure 5 indicate the different mesh schemes and the effect on the hydraulic efficiency respectively.

| No. | N(×10⁶) | ηₘ* |
|-----|---------|------|
| 1   | 0.73    | 84.235 |
| 2   | 1.67    | 85.194 |
| 3   | 3.12    | 85.213 |
| 4   | 3.88    | 85.226 |
| 5   | 4.26    | 85.229 |

Figure 5. Efficiency curve with various elements

3.2.2. Results analysis. Based on the test points and curves stated above, to match the pressure and the discharge specifications with the measurements, relative steady-state simulation is performed at the optimal operating points, of which the results are listed in the table 4.

| NO. | nₑD   | QₑD  | Exp. | CFD | Error |
|-----|-------|------|------|-----|-------|
| 1   | 1.0295 | 0.636 | 93.01 | 86.95 | -6.06 |
| 2   | 1.2108 | 1.04  | 90.68 | 84.82 | -5.86 |
| 3   | 1.2108 | 1.20  | 87.35 | 78.57 | -8.78 |

As can be seen from the table, it is found that there is large difference between the test and CFD results at the similar operating points of which the error ranges from -5.86% to -8.78%, and thus the method applied to the turbine simulation is incompetent as the predicted performance do not agreed with the actual properties.

To dig into the cause, issues to be involved are all taken into account and finally the problem is focused on the interface type instead, by which the individual parts make a whole. Referring to the ANASYS Help [12], stage interface is an alternative to the rotor-stator components, which is most appropriate when the circumferential variation of the flow is of the order of the component pitch. Likewise, for the effect determination, the same operating conditions are simulated for the hydraulic specifications and the results are shown in the table 5.
Through the comparison, the efficiency error at the optimum points is less than 0.5%, of which the least is -0.05% responding to \(n_{ED}\) and \(Q_{ED}\), which is proved to accurately display the performance and further be applied for the hydrodynamics and stability analysis. To make the interface effect clear, the interface locations along the passage are picked out and with the reference of the transient sample, the unique properties of the parameter distribution and data transform are listed in the figure 6 and table 6.

**Table 5. Characteristics of optimal operating points (Stage)**

| NO. | \(n_{ED}\) | \(Q_{ED}\) | Exp. | CFD | Error |
|-----|----------|----------|------|-----|-------|
| 1   | 1.0295   | 0.636    | 93.01| 92.96| -0.05 |
| 2   | 1.2108   | 1.04     | 90.68| 90.34| -0.34 |
| 3   | 1.2108   | 1.20     | 87.35| 87.51| +0.16 |

**Figure 6. Interface locations of the CFD model**

**Table 6. Properties of various interfaces**

| Item | Frozen Rotor Pressure | Frozen Rotor Velocity | Stage Pressure | Stage Velocity | Transient Pressure | Transient Velocity |
|------|-----------------------|-----------------------|----------------|----------------|--------------------|--------------------|
| GV   |                       |                       |                |                |                    |                    |
| RV   |                       |                       |                |                |                    |                    |

From the figures, it can be distinctly distinguished that contrast with transient results, for both steady-state analyses, results from Frozen rotor type brings out more significant variation between the interfaces.
of the same location than that from Stage one, even in the rotor-stator connection like GV-RV and RV-DT when used for relative rotating frame, which leads to improper prediction on the hydraulic performance of the turbine.

3.2.3. Verifications. For the deep verification, this method with stage interface is carried out to obtain the on-cam operating curves attached with different blade angle and with the discrete data points from the CFD simulation, and figure 7 gives the contrast of the experimental and numerical curves.

![Figure 7. On-cam curves of the test and simulation](image)

The curves comparison reveals that with the help of the simulation method, the hydraulic performance of the model turbine unit agrees well with the measurement especially at the low discharge and the optimal conditions while the discharge increase the error appears to be larger, of which the biggest value is about 3.5% that is an acceptable margin error for off-design larger flow, which offers good evidence that the corrected method are more suitable for the tidal turbine unit.

4. Conclusions
In this paper, based on the model test of a low-head tidal turbine, different effect on the performance prediction of the interface type is studied and analyzed and conclusions come out as follows:

By use of CFX tools, compared with Frozen rotor type, a novel model integrated with Stage interface is introduced to be more suitable for the performance check of the low-head tidal turbine, particularly in case that the water level is below 5 m.

As for the low discharge and the optimal points, the hydraulic specifications obtained from CFD simulation agree greatly well with the test results, of which error is even less than 1% while the margin between CFD and test turns out to be 3.5% at the off-design larger discharge, of which the cause needs further research to check the cavitation influence.

Acknowledgement
Special thanks are due to the State Key Program of National Science of China (Grant No. 51439002), the National Natural Science Foundation of China (No. 51479093), the National Key Research and Development Program of China (No. 2017YFC0404200), and the Economy, Trade and Information Commission of Shenzhen Municipality (20141120164551650) for supporting the present work.

References
[1] G.A. Aggidis, E. Luchinskaya, R. Rothschild D. Howard 2010 The costs of small-scale hydro power production: impact on the development of existing potential Renewable. Energy volume
35 pp 2632-2638.

[2] M. Liserre, T. Sauter, J.Y. Hung 2010 Future energy systems integrating renewable energy sources into the smart power grid through industrial electronics *IEEE Ind. Electron Mag.* volume 4 pp 18-37.

[3] Y. Luo, Z. Wang, G. Peng, Y. Xiao, L. Zhai, X. Liu, Q. Zhang 2014 Numerical simulation of a heave-only floating OWC (oscillating water column) device *Energy* volume 76 pp 799-806.

[4] D.D. Prasad, M.R. Ahmed, Y.-H. Lee 2014 Flow and performance characteristics of a direct drive turbine for wave power generation *Ocean. Engineer* volume 81 pp 39-49.

[5] Q. Guo, L. Zhou, Z. Wang 2015 Comparison of BEM-CFD and full rotor geometry simulations for the performance and flow field of a marine current turbine *Renewable Energy* volume 75 pp 640-648.

[6] B.-H. Kim, J. Wata, M.A. Zullah, M.R. Ahmed, Y.-H. Lee 2015 Numerical and experimental studies on the PTO system of a novel floating wave energy converter *Renewable Energy* volume 79 pp 111-121.

[7] Y.Z. Liu, J.S. Chang 2008 Influence of gravity on flow field analysis and hydraulic performance evaluation of bulb turbine *Shuili Xuebao* volume 39 pp 96-102.

[8] Y. Luo, Y. Xiao, Z. Wang 2013 The internal flow pattern analysis of a tidal power turbine operating on bidirectional generation-pumping *IOP Conf.* volume 52 chapter 5.

[9] Y. Luo, Z. Wang, X. Liu, Y. Xiao, C. Chen, H. Wang, J. Yan 2015 Numerical prediction of pressure pulsation for a low head bidirectional tidal bulb turbine *Energy* volume 89 pp 730-738.

[10] P. Lichtneger 2009 Intake flow problems at low-head hydropower, Dresdner Wasserbauliche Mittl. volume 39 pp 259-266.

[11] IEC 60193 1999 *Hydraulic Turbine, Storage Pump and Pump-Turbines-Model Acceptance Test.*

[12] ANSYS Inc. *ANSYS Help Release 16.0* 2015.