Hydrodynamic numerical simulation of diffuser for horizontal axis marine current turbine based on CFD

H Chen 1, D Zhou 2
College of Energy and Electrical, Hohai, Univ. Nanjing, Jiangsu, China

E-mail address: signchx@126.com

Abstract. In order to comprehensively study the hydrodynamic characteristics of diffuser for marine current turbine with a postpositive bulb, a geometric model of the turbine was established. Three-dimensional CFD simulation of turbulent flow was performed based on the incompressible continuity equation, the Navier-Stokes equations and the Spalart-Allmaras turbulence model. The influence of diffuser was calculated and analyzed by numerical results, which were also compared with model test results. Results showed that the numerical results agree fairly well with model test and the maximum error of impeller efficiency and power is 1.5% and 1.8% in the rated water velocity condition, which are less than 5.8% and 5.5% under other cases respectively. The new type marine current turbine with a bulb for erecting motor which is different from regular, and the diffuser can aggregate water flow, raise inlet water velocity more than 3% and efficiency of impeller effectively increased. After diffuser was added, the power coefficient curve rose over the full range, so the high power area became widely, and then remarkably prolonged power time as well as increased generated energy, it is also significant for efficient utilization of marine current energy and environmental pollution remission.

1. Introduction

Marine current energy has rich reserves, relatively mature application technology and a great potential for development in the boundless ocean energy [1]. The horizontal axis marine current turbine (HMCT) can reach a high efficiency and a wide application scope, but the power coefficient is low and power generation time limited as a result of marine current streams have low power density in a long time. Bahaj [2-4] et al use the BEM method for 1.6m HMCT thrust, power and cavitation performance for a water tunnel experiment, then put forward many commercial concept and design schemes. Jing Fengmei [5] et al combined with flume experiment proposed to use a diffuser can significantly reduce starting velocity and improve the power coefficient through CFD method. Zhang Liang [6] et al present that a pair of symmetrical diffusers were installed at the outlet of the dome which can improve the velocity and stability of flow fields in the diffuser. Chen Cunfu [7] et al carried on a research on 2-D diffuser, analyzed the influence of type line and key parameters on hydrodynamic characteristics of HMCT.

Conventional HMCT are similar to wind turbine in the matter of outline structure, and the HMCT with bulb can not only save space on which motor arranged, but also can be used in micro-head water power station, hence the water resources can be efficiently utilized. In the paper, there is a research on hydrodynamic performance of HMCT with bulb and diffuser.
2. Numerical simulation model

2.1. The geometric model parameters
The calculation model structure is shown in figure 1. The diffuser is a rotary body with a circular cross section, and front end is short but after end is contrary, which has a guide water cone inside the front end and a bulb inside the after end, the bulb is provided with a supporting pier, which can play a guiding and supporting role, so that can increase the stability and reliability of plant operation. Coordinate system are defined as follows, the impeller rotor centre is set to \( x=y=z=0 \).

![Figure 1. Geometrical model of the diffuser marine current turbine.](image)

The main geometric parameters: runner diameter \( D_1=0.5\text{m} \), hub diameter \( d_h=0.11\text{m} \), blade number is 5, the length of diffuser \( L_d=0.94\text{m} \), the front section diameter \( D_f=0.54\text{m} \), the after section diameter \( D_b=0.7\text{m} \), the throat diameter \( D_m=0.48\text{m} \), the thickness of the wall \( b=5\text{mm} \), the length, width and depth of the water channel are \( L=5\text{m} \), \( B=1.5\text{m} \), \( H=1.5\text{m} \) respectively. The front and after end are \( 4D_1 \) and \( 6D_1 \) from the channel, and the centre with a distance of \( 1.5D_1 \) and \( 1.5D_1 \) from boundary.

2.2. Mesh generation
The computational domain includes the water channel, the part of water-in of the diffuser, impeller chamber and water-out part of the diffuser. And the impeller chamber is a rotational region, other parts are irrotational regions. Considering the geometrical complexity and the quality of the grid, the author adopt the unstructured grid to improve the grid's adaptability, as well as to ensure that the grid inside the diffuser is rather thin, the grid outside the diffuser being rather thick, as well as the grid near the diffuser's wall local encrypted \(^9\). The verification on grid irrelevance found that there is a small influence on the target research problem when the grid numbers surpass a certain limit. Scheme three is set to be the final one to divide the target models. The different classification scheme based on different grids and the simulation result of impeller efficiency are shown as table 1.

| Scheme | Mesh cells size/10^4 | Impeller | After section of diffuser | Water channel | Total | Impeller efficiency/% |
|--------|----------------------|-----------|----------------------------|---------------|-------|-----------------------|
| I      | 4                    | 32        | 28                         | 76            | 140   | 84.2                  |
| II     | 6                    | 48        | 37                         | 99            | 190   | 84.7                  |
| III    | 8                    | 66        | 50                         | 111           | 235   | 85.0                  |
| IV     | 11                   | 73        | 74                         | 123           | 281   | 85.1                  |

3. Numerical simulation method
### 3.1. Governing equations and turbulent model

Incompressible continuity equation as follows:

\[ \frac{\partial u_i}{\partial x_i} = 0 \]  

(1)

The Navier-Stokes equation is given below:

\[ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_i \partial x_j} \]  

(2)

Where \( u_i \) is the \( i \)th component of the velocity vector, \( \rho \) is the density of the liquid, \( f_i \) is the unit quality of force vector. For the steady flows, \( \frac{\partial u_i}{\partial t} = 0 \).

S-A is a model of single transportation equation with small amount of calculation but brings a good effect on complex boundary problems, especially fitting for problems on external field flow and boundary layer flow with adverse pressure gradient. Therefore, the thesis chooses the S-A turbulence model with high convergence rate but not too high requirement on mesh quality \(^{[10]}\).

### 3.2. Equations discretization and boundary condition

FLUENT6.3 is used as the software platform to complete the numerical simulation work. And the finite volume method is adopted to disperse the controlling system of equations. To ensure the calculate precision, second-order center-difference scheme is adopted to the calculation of the pressure term; second-order upwind scheme is used to the implicit solution of the momentum term and the turbulent viscosity correction; Also, SIMPLEC method is applied on the coupling method of the pressure-velocity, and residual error \( 10^{-5} \) is used as a basis judgment of convergence in the calculation.

The boundary condition is as the following: at the inlet a constant velocity of \( V = 2 \text{m/s} \) was imposed, a rated speed of \( n = 191 \text{r/min} \) was imposed at the impeller and free flow boundary condition was imposed at the outlet. And the surface of the water channel adopts the symmetry boundary Diffuser as the diabatic no-slip wall. What for Impeller is the rotating boundary. And the near wall region adopts the standard law of wall.

### 4. Results and discussions

Refer to Ocean University of China passing the experimental data. Figure 2 illustrates that when rotating angle of blade is 37°, the correlation curve of the impeller efficiency of the ocean current energy water turbine with a diffuser as a result of numerical simulation and model test, and the correlation curve of the power changes as the tip speed ratio changes. When the velocity ratio of the blade top is \( \lambda = 2 \), flow regime performs well, flow regime and the experimental result fit well; and the maximal error of the impeller efficiency is 1.5% and the impeller power is 1.8%. When velocity ratio of the blade top departures, the flow state is rather complex and the numerical simulation is less accurate. Whereas, the variation trends of those two groups is basically corresponding. The maximal errors for the impeller efficiency and power are respectively are 5.8% and 5.5%, which shows that basically the result of the numerical simulation can reflect the performance and the flow field characteristics of water turbine.

The power formula for the ocean current energy impeller is:

\[ P = 0.5 \rho V^3 S C_p \eta \]  

(3)

Where \( \rho \) is the density of ocean, \( V \) is the flow speed, \( S \) is impeller cutting area, \( C_p \) is the power coefficient, \( \eta \) is the transmission efficiency.

The equation (3) shows that the power of impeller is proportional to the cubic velocity. According to the Bates theory, if the velocity increases 3%, the power of impeller can increase 9.3%. Table 2 shows the velocity change of the impeller's entrance when diffusers are added on three different working conditions. According to the diagram, when velocity changes from 1m/s to 2m/s, the velocities are higher as much as 3% or more compared those without diffusers. On this condition, the
velocity can be much easier to reach the minimum self-running speed of the set, which can reduce the running velocity.

**Table.2** Comparison of inlet velocity between with and without diffuser.

| Inlet velocity /m·s⁻¹ | Impeller speed /r·min⁻¹ | Scheme       | Inlet velocity of impeller /m·s⁻¹ | Speed improvement/% |
|------------------------|-------------------------|--------------|----------------------------------|---------------------|
| 2                      | 191                     | Without diffuser | 1.70                            | 3.5                 |
| 1.5                    | 153                     | With diffuser   | 1.76                            | 3.9                 |
| 1                      | 115                     | Without diffuser | 1.28                            | 3.4                 |
|                        |                         | With diffuser   | 1.33                            |                     |

Diffuser can not only increase the conveyance capacity of water turbine, but also can largely increase output power of the water turbine. Figure 3 is the changing curve of the impeller power of the water turbine and the effective axial force of the impeller axis along with the velocity ratio of the blade top.

According to the figure, curves of power of the water turbine and the thrust force of the impeller rise obviously. When rated velocity ratio of the blade top reaches to $\lambda=2$, the power of the water turbine without impellers is 300w, the power of diffuser water turbine is 550W, their power increasing as twice as much as before. Within the water turbine operation range, the power range of the water turbine with diffusers increases, which means high power area in the water turbine with diffuser increases, and the optimum velocity ratio of the blade top increases from 2 times to 2.5 times. Meanwhile, the thrust force of the impeller on the water turbine increases. When rated velocity ratio of the blade top is $\lambda=2$, the thrust force increases from 275N to 415N, increasing 50% as much as before. Therefore, the strength design and proofreading of impeller are an important part of the ocean current energy water turbine.

The influence on the water turbine's performance is the changing to the inlet conditions of its blade top. Figure 4(a) shows that there is no obstructing from diffuser in a water turbine without diffuser. When the speed on the same axial position is higher than that of a water turbine with diffuser, decreases from both sides of the axis, parabolic type, but the distribution of the radial velocity gradient is large, which means the flow is not stable before entering the impeller, and the speed changes greatly, the diffuser can provide a rather good steady flow. While the obstruction of the wall, the water gathered in the hood, and changes little in speed, which provides a good inflow condition. Along the flow direction, when the water turbine without a diffuser impels itself to the impeller chamber, the
water velocity reduces on the whole. While water turbine with diffuser is on the country, which allows the water velocity of the entrance on the laryngeal part of the diffuser reaches its highest level.

(a). The impact on speed radial distribution of diffuser inlet.

(b). The impact on speed radial distribution of impeller outlet.

**Figure 4.** The diffuser impact on external characteristic of turbine (V=2m/s).

According to figure 4(b), the impeller entrance flow velocity of the water turbine with diffuser overalls than those without diffuser. As turbine blades due to the singular number, the center blades longitudinal section do not exist symmetry at the same time, water being stirred. That is why the flow velocities of both inlet and outlet have asymmetrical distribution along the axis to the two sides. On the main areas beyond the minimum section of the water turbine with diffuser, the block from the diffuser makes the flow velocity decreases radially from the wall surface, reaching to its minimum at the axis and to its maximum at the blade tip, which is due to the rotation of the impeller blades makes the outer edge of the wing reaches its maximum circumferential velocity, and the blades rotation drives flow of the surrounding fluid faster.

Figure 5 is the streamline and velocity contour in horizontal axis section of turbine. As there is no slip-area on the diffuser wall, a thin layer forms on the wall making a boundary layer. The water velocity reduces sharply until to zero on the wall, which conforms to the boundary layer theory. When the water reaches to the front part of the diffuser, it changes the flow direction for the restraints of the diffuser. Therefore there will be three zero-speed stagnation points on the both sides of diffuser as well as the head of the hydro-cone. On the back part of the diffuser is the wake zone. As the mutation of the water crossing section on the back of the bulb, there are large back flowing areas on the back, which will have an influence on the recovering of the water velocity of the back part.

**Figure 5.** Velocity contours and streamlines at vertical section of diffuser turbine.
As the disturbing from the turbine without diffusers to the blades in opening area, streamline of the entire disk radially offset from the center to the sides. Flow velocity of impeller changes great. There is a rather large back flowing area on the neck of the bulb. Therefore, the blades on up and downstream disc surface are stress unevenly. After the installation of diffuser, the restraints to the water flow from the diffuser make flow forced to gather in the diffuser area, streamline balances on the up and downstream disc surface, flow velocity changes stable and blades carry a balanced stress on both up and down streams. Meanwhile, there is a rather large export circulation due to the blades rotation of the blade exist part. The diffuser reduces the circular velocity of the exit flow and prevents it flowing to the outside, making the water flows along the bulb wall better, improving the swirl and backflow area of the neck part on the bulb, making the flow more gathered flowing along the axis.

In the runner chamber, blades can have an influence on the out edge of impeller of turbine without impellers flow lines obvious deflecting outward, water gushing out in a radial outward rotation of the impeller and "throwing water" outward. As figure 6 shows that this "throwing water" phenomenon leads to inconsistent impeller import and export traffic, causing loss of flow, so that the impeller and export discontinuity. After adding the diffuser, as the blocking of the wall, "dumping water" no longer exist and the two edges of the impeller are stressed proximately parallel. And the set are in stable operation.

![Figure 6](image.png)

(a). The relative velocity distribution of blade surface that without diffuser.  
(b). The relative velocity distribution of blade surface that with diffuser.

**Figure 6.** The relative velocity distribution of blade surface ($V=2\text{m/s}$).

5. Conclusions

Taking the marine current energy water turbine with added postpositional bulb as the object of study, by adopting numerical simulation combined with model experiment to calculate and analysis the turbine with a diffuser, the thesis verify the reliability and the accuracy of the numerical simulation method and conclusion is made as the following:

1. The diffuser has big effect to the promotion of hydraulic performance. The velocity of impeller inlet has increased more than 3% due to the strong aggregate and speed-up effect, then the impeller power increased nearly doubled, and the high power area became widely. At the same time, the larger inlet velocity will decrease the starting velocity of unit, thereby remarkably prolonged power time as well as increased generated energy.

2. The diffuser plays an important role to improving the flow field. The water flow direction changed because of diffuser which will have more fluid suction to the hood, and the streamlines will
be more concentrated. The diffuser has not only reduced the region of reflow in the bulb neck, but also stopped "throwing water" phenomenon in the impeller chamber on account of high impeller tip rotation speed, and then the stability in operation of the turbine unit would be well guaranteed.

References

[1] Xu X T 2012 Design and characteristic analysis on blades of horizontal axis marine turbine's mechanics. Zhejiang University. Hangzhou. Zhejiang

[2] Batten W.M.J, Bahaj A.S, Molland A.F, etal 2007 Experimentally validated numerical method for the hydrodynamic design of horizontal axis tidal turbines Ocean.Eng 34 1013-20

[3] Batten W.M.J, Bahaj A.S, Molland A.F, etal 2006 Hydrodynamics of marine current turbines Renewable.Energy 31 249-56

[4] Batten W.M.J, Bahaj A.S, Molland A.F, etal 2008 The prediction of the hydrodynamic performance of marine current turbines Renewable.Energy 33 1085-96

[5] Jing F M, Zhang L, Zhang P Y and Xiao L J 2012 Study on an accelerated diffuser of tidal current power generation J. Harbin.Eng.Univ.33 409-13

[6] Zhang L, Sun K, Luo Q J 2007 Hydrodynamic design of diversion cover for a tidal stream hydro turbine J. Harbin.Eng.Univ. 28 734-37

[7] Chen C F,Wang S J, Yuan P and Tan J Z 2011 Hydrodynamic Numerical simulation of diffuser for horizontal axis tidal-current energy turbine Proc. of The Chinese Renewable Energy Institute Academic Conference.

[8] Liu B 2007 Study on the Hydrodynamic Performance of Vertical-axis turbine with demos Harbin Engineering University. Harbin. Heilongjiang

[9] Xin X P, Shao X M, Deng J and Li W 2011 Hydrodynamic performance prediction of marine current turbine with dual rotor in tandem arrangement J. Zhejiang. Univ (Eng Sic) 45 1227-31

[10] Bao Y, Zhou D, Huang C et al 2011 Numerical prediction of aerodynamic characteristics of prismatic cylinder by finite element method with spalart-allmaras turbulence model Comput. Struct., 89 25-338