Applied Mechanics and Materials Vols. 672-674 (2014) pp 386-391
© (2014) Trans Tech Publications, Switzerland
doi:10.4028/www.scientific.net/AMM.672-674.386
Submitted: 06.08.2014
Accepted: 09.08.2014

Efficiency and performance analysis of tidal current energy turbine basing on the unidirectional fluid-structure interaction

LIU Xue-feng[1, a], WANG Jin-bao[2,b], TIAN Mei-ling[2,c] and TANG Zhi-bo[2,d]

[1]School of ship and ocean engineering, Zhejiang Ocean University,Zhoushan,Zhejiang,316022,China

[2]School of Shipping and Ports Architecture Engineering, Zhejiang Ocean University,Zhoushan,Zhejiang,China

[a]speaknow@163.com, [b]jinbaow@zjou.edu.cn, [c]tian_ml@126.com, [d]zhibo_tang@zjou.edu.cn

Keywords: Horizontal Axis Tidal Current Energy Turbine; Blades; Unidirectional fluid-structure interaction;

Abstract: As the key components of a horizontal axis tidal current energy(HATCE) turbine, the blades will be affected by the force of the fluid when the turbine is working, which also results in a possible effect on the safety and stability of the tidal current energy turbine. Thus, both the structural performance and energy-catching efficiency of HATCE turbine should be paid equal attention. In this study, basing on the Workbench, the energy-catching efficiency and structure performance of the designed HATCE turbine with stainless steel and structural steel at the different current speeds are comparatively studied using unidirectional FSI analysis method. It can be concluded that the output power of the turbine is lower at a low current speed but its energy-catching efficiency is higher and vise versa. As a result of structure performance analysis, the designed turbine has adequate safety under all loaded conditions. Thus, the designed turbine models are available.

Introduction

As the key components of HATCE turbine, the performance of the blades will have an effect on its efficiency and stability when it's working. A few researchers have studied the tidal current energy turbine. Chul hee Jo et al. gave the design procedure for a 300 kW tidal current turbine blade and studied the performances of the 3D turbine model by the CFD method [1]. In Ref.[2], 1-MW class tidal current turbine was proposed and the performance was studied numerically by BEM theory and CFD. B.S. Kim researched the performance and structural safety of 50 kW ocean current turbine rotor by CFD simulation and unidirectional FSI analysis[3]. It can be found that many works care more about the turbine's power coefficient than its structural safety. However, the turbine's efficiency and strength should be taken into account at the same time and it can be achieved by fluid-structure interaction analysis.

Fluid-structure interaction analysis is used to study the interaction between the structure field and fluid field, that is to say, Structure will deform or move under the action of fluid loads and the deformation or movement of structure will affect the distribution and size of fluid loads[4]. Fluid-structure interaction(FSI) can be divided into unidirectional and bidirectional ones. Unidirectional FSI is usually used when the deformation of structure has little effect on the fluid field, in which fluid and structure analyses need to be conducted only once, respectively. For bidirectional FSI, it is used when the influence of the structure's deformation on the fluid field cannot be ignored, and fluid and structure analyses need to be conducted alternately.

In this study, power coefficient calculation and strength analysis of a small horizontal axis tidal current energy turbine are conducted by using of unidirectional fluid-structure interaction. This study is carried out via Workbench module in the ANSYS software, in which ANSYS CFX and Static Structural programs are integrated. Steady numerical simulation can be performed using CFX to get the loads in the fluid field. Then the power coefficient and the loads transferred from fluid field to the structure field can be calculated. Finally, strength analysis can be performed using Static Structure program and stress distributions of the blades can also be obtained.

CFD analysis of the HATCE turbine

The configurations of the selected HATCE turbine are shown in **Table 1**.

Table 1 Configurations of the TCE turbine

Item	Value	Item	Value
Design power	30 *W*	Design current velocity(m/s)	0.8 *m/s*
Design tip speed ratio	3.0	Maximum current velocity	5.0 *m/s*
Turbine radius	0.3 *m*	Rotating speed	8 *rad/s*
Blade number	5		

The turbine model was built using Solidworks software and shown in **Fig. 1**. According to Ref.[5], the numerical simulation was performed with the MRF model using the ANSYS CFX. The cylindrical computational domain can be divided into stationary field and rotating field. And the size of the domain was marked in the **Fig. 2**. The turbine model should be contained in the rotating field and its rotating speed is about 76 *rev/min*.

Fig. 1 The TCE turbine model

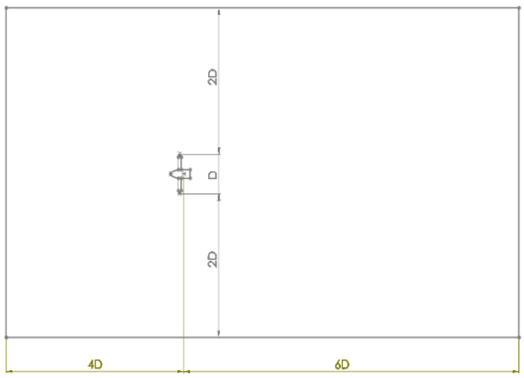
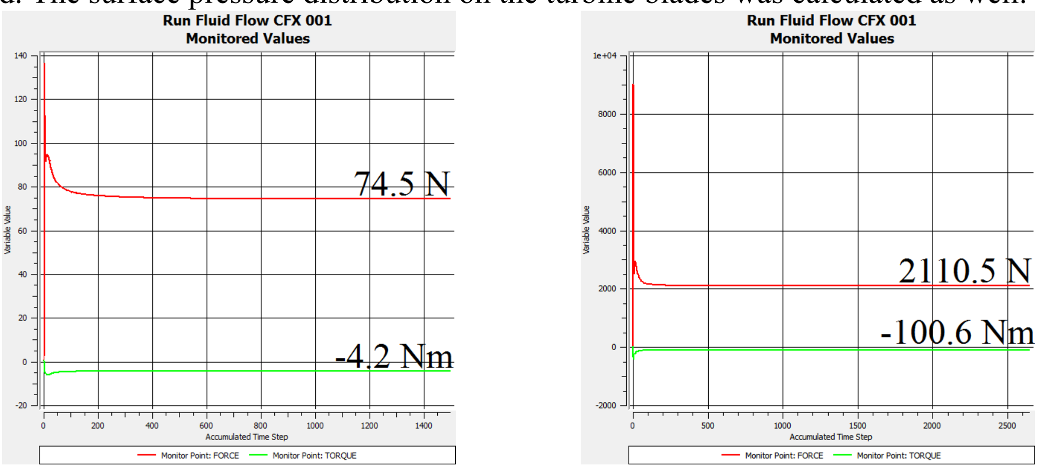


Fig. 2 The computational domain

After all the model was built well, the mesh generation was performed using the ANSYS ICEM CFD. Subsequently, the boundary conditions were set. The inlet and outlet boundary were adopted. The normal speed at the inlet is known and the relative average pressure over the whole outlet is 0 *Pa*. The turbine surface was set as wall and rotating with the rotating field. The rotating speed is also known and the rotating direction is anticlockwise in **Fig. 1**. In the simulation process, the SST turbulence model was adopted, which can accurately predict size and onset of flow separation generated by adverse pressure gradient. And the thrust and torque of the turbine blades were monitored. The surface pressure distribution on the turbine blades was calculated as well.



(a) At the design current speed of 0.8 *m/s* **(b)** At the maximum current speed of 5 *m/s*

Fig. 3 The monitored values of thrust and torque

Fig. 3(a) and **Fig. 3(b)** show the monitored values of thrust and torque at the design and maximum current velocity, respectively.

Strength analysis of the HATCE turbine by unidirectional FSI

Unidirectional FSI analysis

In this study, the pressure applied by the fluid field is used as the initial load condition on the turbine structure in unidirectional FSI analysis, and then stress and strain of the turbine structure can be obtained. Specially, the turbine surface pressure load from CFD analysis is mapped directly on the turbine structure using interpolation method. And the mapped surface pressure load can be converted to forces in x, y, and z directions, respectively.

The forces before and after mapping at different current velocity are shown in **Table 2**.

Table 2 The forces before and after mapping

Tidal current velocity (m/s)	Force in each direction (N)	Before mapping (CFD force)	After mapping (structure force)
0.8(design velocity)	Fx	74.5460	74.3770
	Fy	-0.1214	-0.1534
	Fz	-0.0279	-0.0062
5(maximum velocity)	Fx	2110.5	2091.6
	Fy	-3.1601	-3.4984
	Fz	-5.3015	-5.1879

From the **Table 2**, it can be seen there is a little difference between the force values before and after mapping. The possible reasons result from the following analysis. On the FSI interface between fluid field and structure field, the mesh nodes do not coordinate completely with each other, and there is a little offset. Through mesh nodes on the FSI interface, the CFD pressure is directly transferred to the structure field via the interpolation method, and thus it will result in a little difference of the forces before and after mapping

Strength analysis by unidirectional FSI

Here the stainless steel and structural steel are chosen as the material of the turbine and are compared. Their material properties are listed in the **Table 3**.

Table 3 The properties of selected material

Item	ρ (*kg/m3*)	E (*Pa*)	μ	Sy (*Mpa*)	Su (*Mpa*)
Structural steel	7850	2e11	0.3	250	460
Stainless steel	7750	1.93e11	0.31	207	586

HATCE turbine structure model is meshed with 152560 elements and is shown in Fig. 4, in which the tetrahedron element is adopted as the mesh type to fit the complex shape and deformation of the turbine. Then the constraints, boundary and load conditions are applied. The rotating velocity is 8 rad/s anticlockwise. Finally, the stress and total deformation of the turbine can be calculated when CFD pressure is mapped onto the turbine structure.

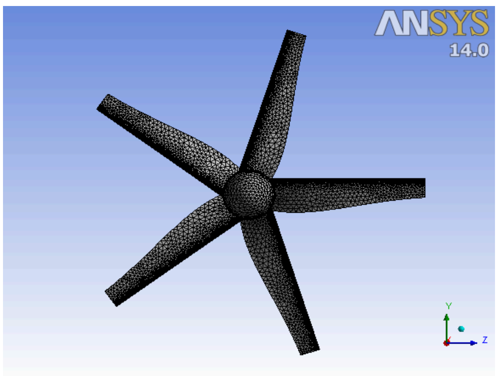


Fig. 4 The mesh model of the turbine

Results and discussions

Power output evaluation

From **Fig. 3**, it can be known that the torques acting on the turbine blades are 4.2 *Nm* and 100.6 *Nm* at the tidal current speed of 0.8 *m/*s and 5 *m/s*, respectively. Then, the power of the turbine can be calculated by the definition as

$$P = T\omega \tag{1}$$

where T is the torque value, and ω is the rotating velocity. Here, the output power of the turbine at the two kind of tidal current velocities are 33.6 *W* and 804.8 *W*, respectively. It can be seen that the power at the designed speed is bigger than the designed power. Then a multiplier η less than 1 is introduced which represents some resistance to be overcome when the turbine is rotating. The resultant power satisfies the design requirements. It should be noted that the torque value aforementioned is one acting on the turbine blades not the whole turbine.

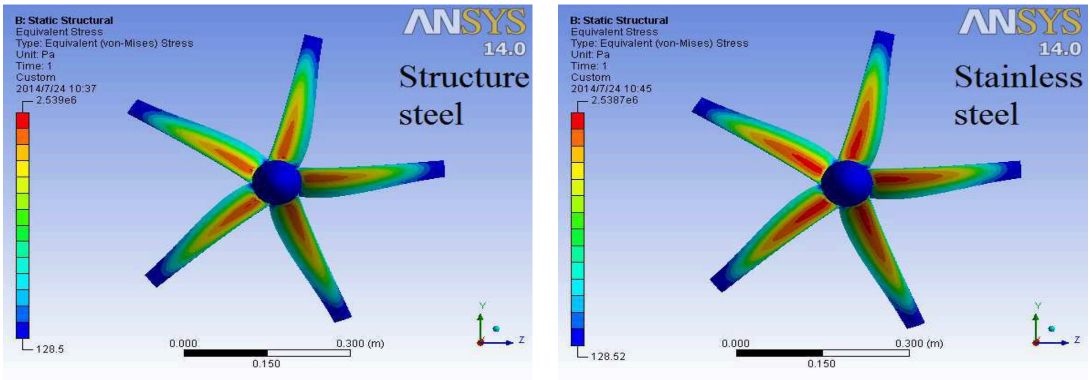
Furthermore, the energy-catching efficiency C_P of the turbine can be obtained with the definition as

$$C_P = P/(0.5\pi\rho R^2 V_\infty^3) \tag{2}$$

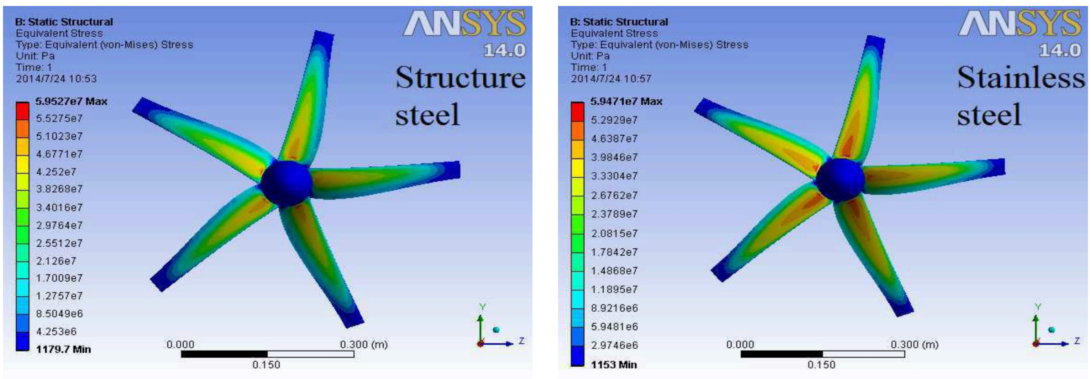
where ρ is the density of water, R the radius of the turbine and V_∞ the tidal current velocity. The energy-catching coefficients are 0.46 and 0.05 at the design speed and maximum speed, respectively. Obviously, the output power of the turbine is higher at a higher fluid speed but its efficiency is lower. That is to say, the efficiency is higher at a low fluid speed while the output power is lower.

Strength evaluation

The stress and total deformation are obtained from the structural safety calculation and shown in **Fig. 5** and **Fig. 6**, respectively.



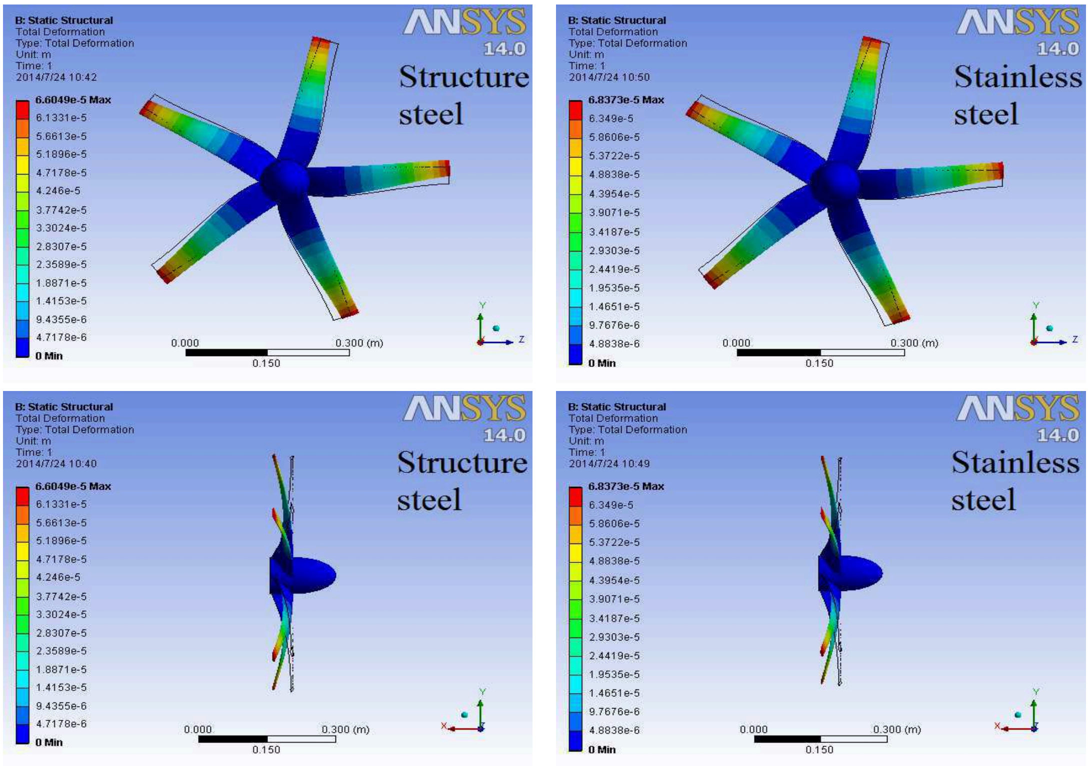
(a) At the speed of 0.8 *m/s*



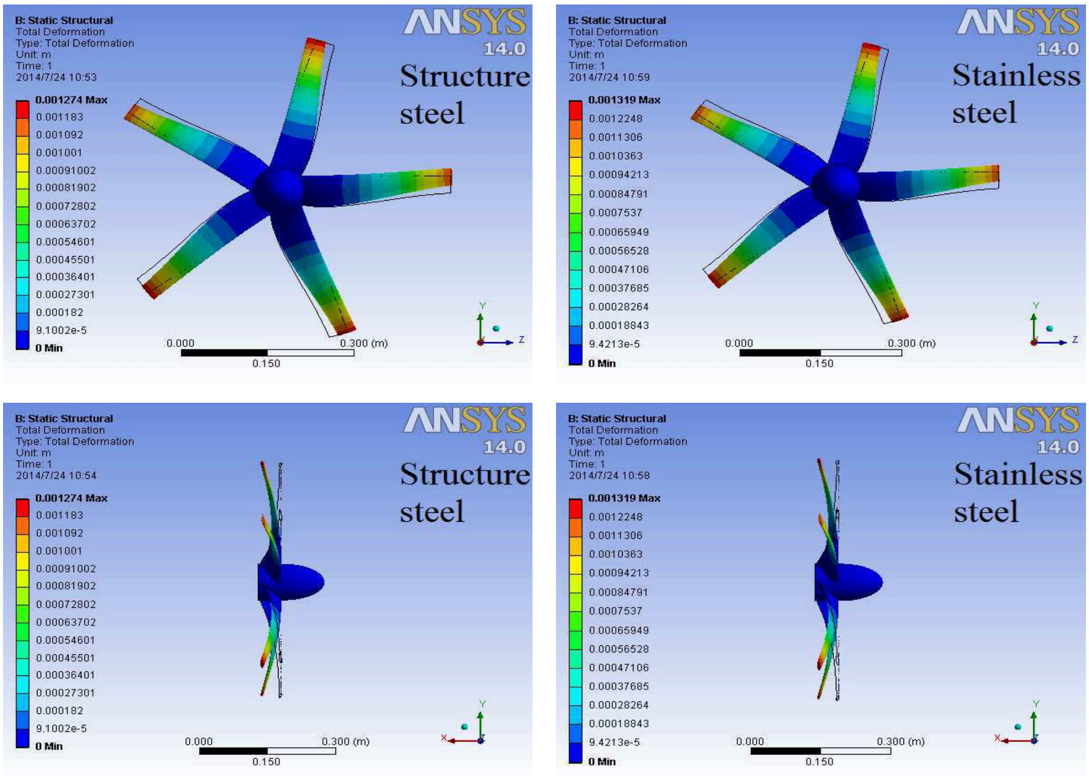
(b) At the speed of 5 *m/s*

Fig. 5 The stress distribution of the turbine structure

In the **Fig. 5(a)** and **Fig. 5(b)**, the maximum von-Mises stress exists at the blade root. The stress is 2.54 *Mpa* for the two kind of material at the design current speed of 0.8 *m/s*. At the maximum current of 5 *m/s*, the stress are about 59.53 *Mpa* and 59.47 *Mpa* for the structure steel and stainless steel, respectively. It can be found that the maximum von-Mises stresses for two kinds of materials at the different current speeds are smaller than their corresponding yield strength, so the turbine structure is safe and can meet the strength requirements.



(a) At the speed of 0.8 *m/s*



(b) At the speed of 5 *m/s*

Fig. 6 The total deformation of the turbine structure

From the **Fig. 6**, it can be shown that the turbine structures deform in the axial and circumferential directions due to the thrust and torque acting on it and the maximum deformation exists at the blade tip. For the material of structure steel, the maximum total deformations are 0.066 *mm* and 1.274 *mm* at the speed of 0.8 *m/s* and 5.0 *m/s*, respectively. While for the material of stainless steel, the maximum total deformations are 0.068 *mm* and 1.319 *mm* at the two different speeds. The deformation quantities of the turbine structure under different conditions are very small. On the other hand, the turbine hub has almost no deformation and the stress here is also very small. The turbine structure also satisfies the rigid requirement. Thus, the present designed turbine structures are available and feasible.

Conclusions

In this study, the energy-catching efficiency and structure performance of two kinds of materials of HATCE turbine at the designed and maximum current speed are studied using unidirectional FSI analysis method. From the CFD analysis, it can be concluded that the output powers of the turbine at the designed and maximum current speed are 33.6 *W* and 100.8 *W*, respectively. However, the energy-catching coefficients are 0.46 and 0.05, respectively. It can be found that the output power of the turbine is lower at a low current speed but its efficiency is higher and vise versa. As a result of structure performance analysis, the turbine has adequate safety under all different conditions. Thus, the turbine model meets the design requirements.

Acknowledgment

This work was supported by the Key Program of Zhejiang Ocean University.

Reference

[1] Jo C, Yim J, Lee K, et al. Performance of horizontal axis tidal current turbine by blade configuration[J]. Renewable Energy, 2012, 42: 195-206.

[2] Moon-Chan Kim, In-Rok Do, et al. Numerical Investigation on the Performance of 1-MW Class Tidal Current Horizontal Axis Turbines[C]. Proceedings of the Twenty-second International Offshore and Polar Engineering Conference, 2012.

[3] B S Kim, S Y Bae, W J Kim, S L Lee and M K Kim. A study on the design assessment of 50kW ocean current turbine using fluid structure interaction analysis[C]. 26th IAHR Symposium on Hydraulic Machinery and Systems. IOP Conference Series: Earth and Environmental Science 15 (2012) 042037. IOP Publishing, 2012.

[4] Tao Lou. Numerical simulation of fluid-structure interaction basing on the ANSYS[D]. Lanzhou University, 2008:1-2.(In Chinese)

[5] Cunfu Chen. Study on Blade Optimization and Hydrodynamic Performance of Horizontal Axis Marine Current Turbine[D]. Qingdao: Ocean University of China, 2012.(In Chinese)