Numerical Simulation of An Axial Flow Fan with Proper Aft Fins for Floating Wind Turbines Model Test

Yu Long[1,2,3], *Huang Chao*[1,2,3*], *Zhao Yongsheng*[1,2,3], *He Yanping*[1,2,3]

1. State Key Laboratory of Ocean Engineering, Shanghai Jiao Tong University, Shanghai 200240, P. R. China;
2. Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration(CISSE), Shanghai 200240, P. R. China;
3. School of Naval Architecture, Ocean & Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, P. R. China

(Received 18 October 2015; revised 20 February 2016; accepted 5 March 2016)

Abstract: It is difficult to generate homogeneous wind in model test for floating wind turbines, especially when using array of axial fans. We design and compare proper fins behind the blades for an axial fan to produce anti-twist flow based on the computational fluid dynamics (CFD). The result is promising and proves that suitable designed fins can be an effective solution.

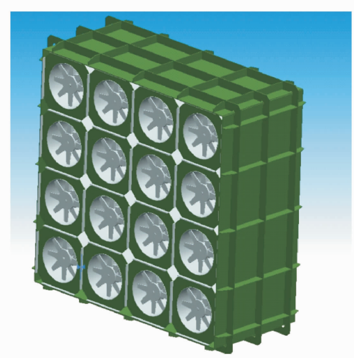
Key words: homogeneous flow; fins; anti-twist flow; CFD; full scale

CLC number: U661 **Document code**: A **Article ID**: 1005-1120(2016)06-0754-07

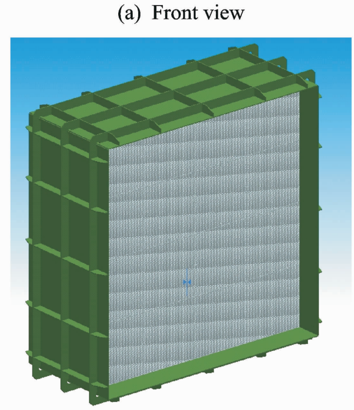
0 Introduction

Generating homogenous wind for model test of floating wind turbines is essential but difficult. The homogenous wind inflow determines the aerodynamic performance of the wind turbines[1-2]. Axial fans are ordinary devices for generating wind, but they inevitably generate a twisted flow, which introduces spatial inhomogeneity and high turbulence level. Therefore, the alternative centrifuge air producer has been applied[3]. The researchers tried various instruments, like diffuser, honey comb, screen etc. to reduce the twist. In the model test for a TLP floating wind turbine in the deep water basin of Shanghai Jiao Tong University, an array of axial fans have been settled for wind inflow generations.

The axial fan array is composed of 4×4 axial fans to generate an adequate air inflow of the wind turbine, as shown in Fig. 1. The honey comb and screen are installed at the back of the fans with a circle transform to rectangle structure, as shown in Fig. 2. The axial fans are shrouded with a certain length and mounted to rectangle screen.



(a) Front view



(b) Back view

Fig. 1 The axial fan array

* Corresponding author, E-mail address: hc110@sjtu. edu. cn.

How to cite this article: Yu Long, Huang Chao, Zhao Yongsheng, et al. Numerical simulation of an axial flow fan with proper aft fins for floating wind turbines model test[J]. Trans. Nanjing Univ. Aero. Astro., 2016,33(6):754-760.
http://dx. doi. org/10. 16356/j. 1005-1120. 2016. 06. 754

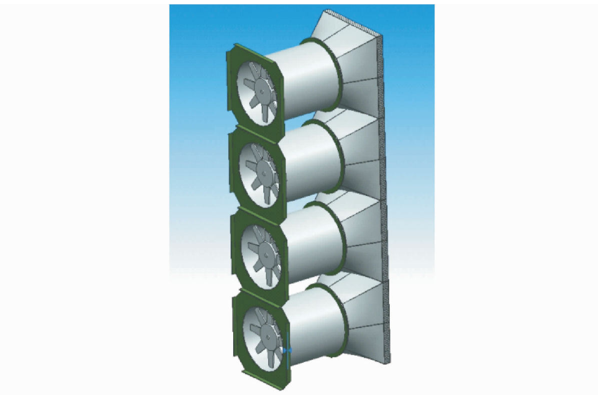


Fig. 2 Connections of the axial fans

A typical axial fan have multiple blades and a shroud around the blades(Fig. 3).

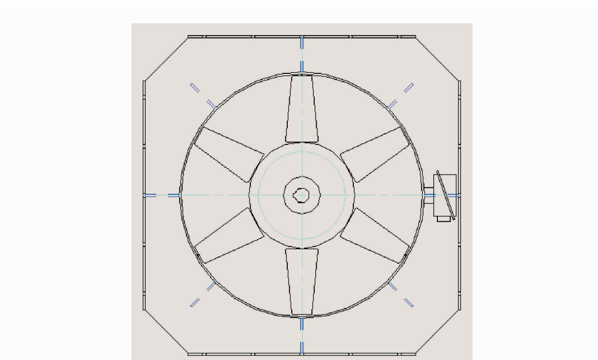


Fig. 3 Details of an axial fan

The main parameters of the axial fans are listed in the Table 1.

Table 1 The main parameters for the axial fans

Item	Value
Overall length / mm	500
Diameter / mm	570
Motor length / mm	315.5
Motor diameter / mm	224
Hub of the blade / mm	250
Number of the blades	6
Gap of the tip / mm	5
Blade axis to inlet / mm	107
Motor fore end of inlet / mm	157

1 Theoretical Background

The combination of a propulsor and the fins is a general technique for minimizing rotational energy loss for ship propulsion[4-5].

The governing equations of continuity and momentum are listed as follows[6]

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}) = 0$$

$$\frac{\partial (\rho \boldsymbol{U})}{\partial t} + \nabla \cdot (\rho \boldsymbol{U} \otimes \boldsymbol{U}) = -\nabla p + \nabla \cdot \tau + S_M$$

where $\nabla = \left[\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}\right]$, S_M is the momentum source, $\boldsymbol{U}$ the vector of velocity, ρ the density, and t the time step.

In the analysis, ideal gas has been applied as fluid material

$$\rho = \frac{w P_{abs}}{R_0 T}, dh = C_p dT, C_p = C_p(T)$$

where w is the molecular weight, P_{abs} is the absolute pressure, R_0 the universal gas constant, and T the temperature.

2 Analysis Without Fins

2.1 Geometry

The typical feature of this kind of axial fan is that a shroud surrounds the rotor as a wall, with a small gap between the blade tip and the shroud, modelled as Fig. 4.

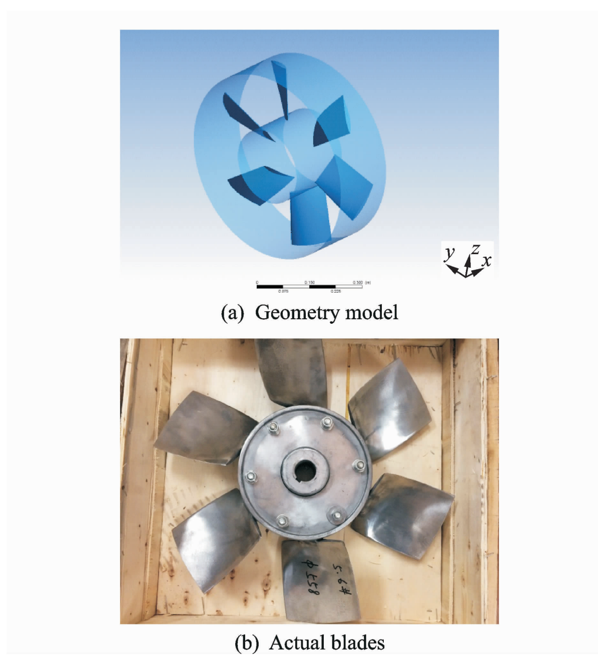


Fig. 4 Geometry comparison of an axial fan and actual blades

2.2 Mesh and boundary conditions

A single passage with only one blade has been modelled in Ansys. CFX, with periodical boundary conditions to reduce the computation cost (Fig. 5). The velocity inlet and pressure outlet have been used. Structured grids are used for all the domains. The inlet flow zone is 4 times the length of blade diameter while the wake zone is 8 times of that and the outer circle range is 4 times of that, which is also proper for ducted propeller analysis[7] and proved to be large enough by later calculations. The rotational block consists of 104 730 hexahedra elements and total grids are 585 980 hexahedra elements.

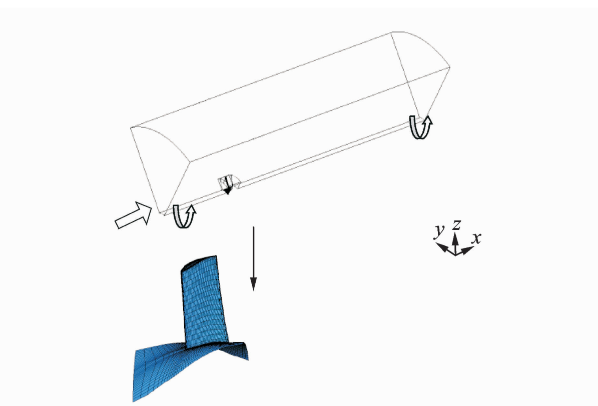


Fig. 5 Periodical boundary conditions and rotor mesh

2.3 Validation for the original single-fan rotor

At the beginning of the analysis, a validation calculation is conducted for the original single rotor. Steady calculation has been carried out since it has adequate accuracy and reasonable computational cost compared with unsteady analysis[8].

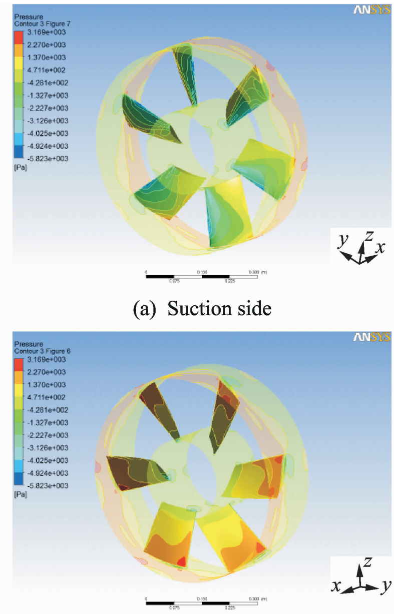
According to the aerodynamic performance of the axial fans[9], the efficiency and the power are calculated and listed in Table 2.

The differences between test data and numerical calculations are small enough for further research.

The shroud has a big influence on the blade pressure distribution due to the fact that the blade tip vortex can induce velocity loss, as shown in Fig. 6.

Table 2 Validation of design point

Item	Test data	Calculation	Differences
Power / kW	6.512	6.098	6.78
Efficiency / %	73.20	67.00	6.20



(a) Suction side

(b) Pressure side

Fig. 6 Pressure contour of the blade, shroud and hub

The large hub diameter has a negative impact on the performance by inducing a vortex in the slipstream(Fig. 7).

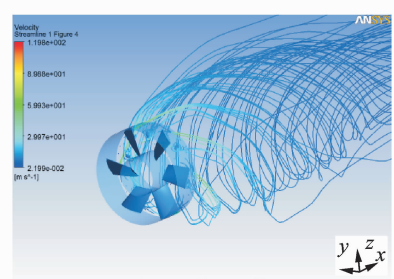


Fig. 7 Streamlines after the rotor

The spiral lines are obvious due to the rotation of the rotor, which causes the disturbed airflow behind the rotor disc. The aft fins will be brought out for weakening the twist.

3 Analysis with Fins

For saving the investment cost, the motor, the rotor and the shroud length are fixed. Then in computation, the whole calculation domain has been divided into five parts: Inlet domain, rotor/blade domain, fins in the wake domain, outlet domain and outer fluid domain. Only the fins in the wake domain will change during the analysis. Generally, the aft fins are described by hydrofoil and some boundary parameters[10]. The shape of the fin is developed from NACA0015 foil geometry.

For a given radius plane section, as shown in Fig. 8, the distance between the rotor and the fin L, the chord length of the rotor C_r and the thickness of the rotor t_r are kept unchanged because the rotor and the shroud are already determined. The chord length of the fin C_s, the inflow angle of the fin φ and the thickness t_s are selected as design variables.

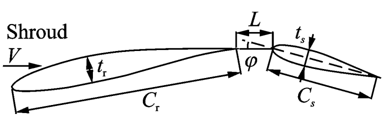


Fig. 8 Rotor and fin design variables in a given radius section

The rotor with 6 fins is modelled and shown in Fig. 9.

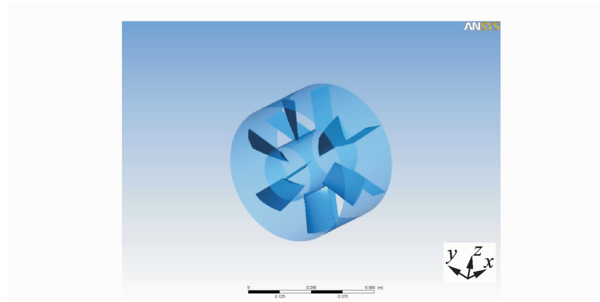


Fig. 9 Rotor and fins

After a preliminary selection procedure, three models of fins have been designed(Fig. 10).

Then the rotor with fin models are computed with the same inflow mass flow rate.

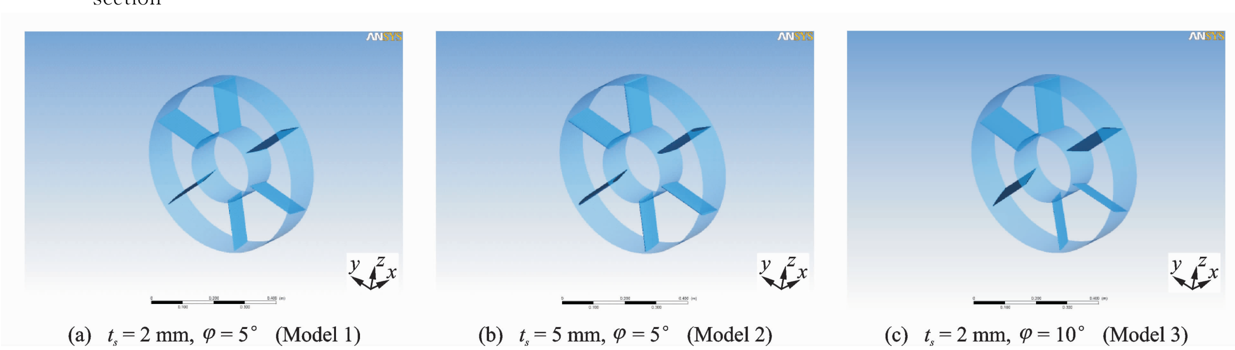


Fig. 10 Geometry model of three different fins

4 Results and Discussion

4.1 General analysis

All the models are computed by the shear stress transport (SST) turbulence model[11].

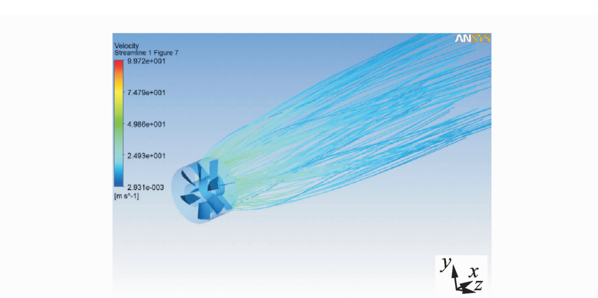


Fig. 11 Streamlines after the fin for model 1

The streamlines are much more uniform than the single rotor (Fig. 7) because the hub vortex has been crippled. The fins effectively divided big vortex at the hub and tip vortex into several smaller ones which can be neutralized at some spaces.

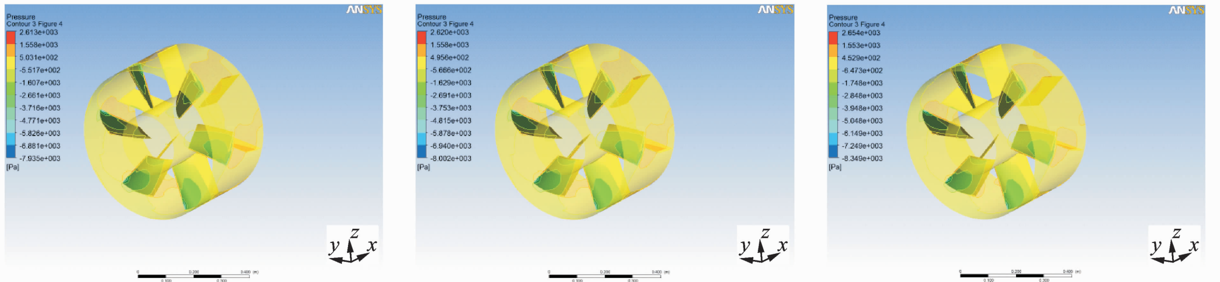
With the same inflow mass rate and geometry diameters, the fins model can save the total power about 11% than the original single rotor (Table 3). This is because the fins collect the flow and absorb some energy loss in the slipstream by preventing the flow twist.

Table 3 Comparison of fin effect

	Single	Model 1	Model 2	Model 3
Power / kW	6.1	5.43	5.45	5.32

4.2 Blade pressure analysis

According to Fig. 12, the existence of fins decrease rotor blade pressure, compared to single rotor (Fig. 6).



(a) $t_s = 2$ mm, $\varphi = 5°$ (Model 1) (b) $t_s = 5$ mm, $\varphi = 5°$ (Model 2) (c) $t_s = 2$ mm, $\varphi = 10°$ (Model 3)

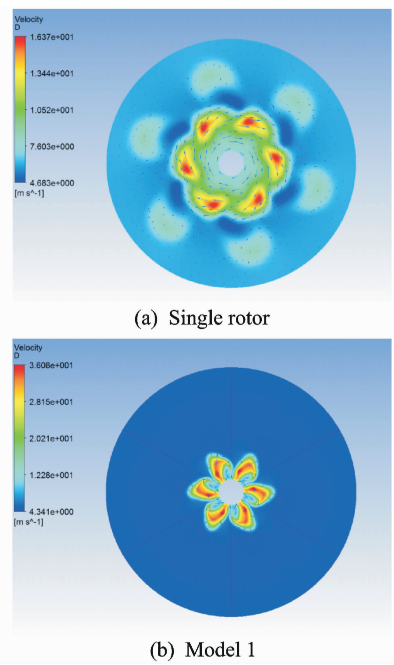
Fig. 12 Pressure contours of the rotor blades, fins and hubs for different fin models

The total pressure of the blades with fins drops obviously compared to that of the single rotor, which means the velocity of the airflow increases based on Bernoulli law. Among the three models, the inflow angle and thickness of the fins have significent impact on the blade pressure of rotors and stators. The thickness increment of the fins slightly increases the blade pressure while the inflow angle of the fin can induce a larger pressure.

4.3 Slipstream analysis

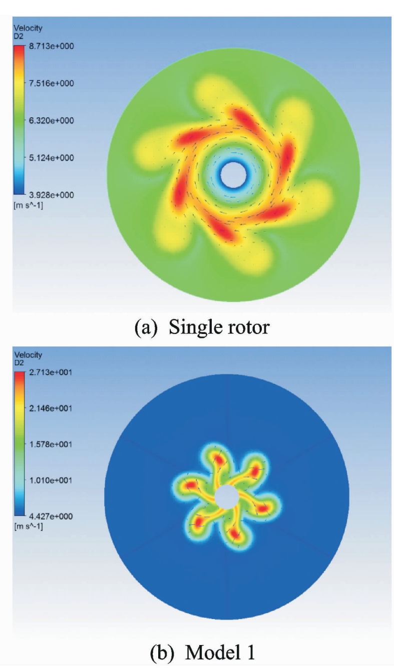
Since the purpose of this research is to generate homogenous flow by the aft fins, the slipstream of the axial fans is also investigated. Velocity is the direct index for uniformity of the flow. Velocity contours of three sections in the slipstream with 1 diameter (1D), 2 diameters (2D), 3 diameters (3D) and 6 diameters (6D) behind rotational blades are chosen for both single rotor and rotor with fins, as shown in Fig. 13, Fig. 14, Fig. 15, and Fig. 16 respectively. The section diameter is 2 times of shroud diameter. The arrows indicate the strength of the circumferential velocity, as twist flow strength indications.

At 1D behind the blades, the values of velocity of the single rotor are much smaller than those of the rotor with fins. This is because the fins block and collect the passing through airflow and reduce the hub vortex by increasing the velocity at blade main body. The velocity of the single rotor has a gradually rising from the hub to tip. The circumferential velocities at hub domain in single rotor case are larger than that of rotors with fins, which also prove the reduction of hub vortex.



(a) Single rotor (b) Model 1

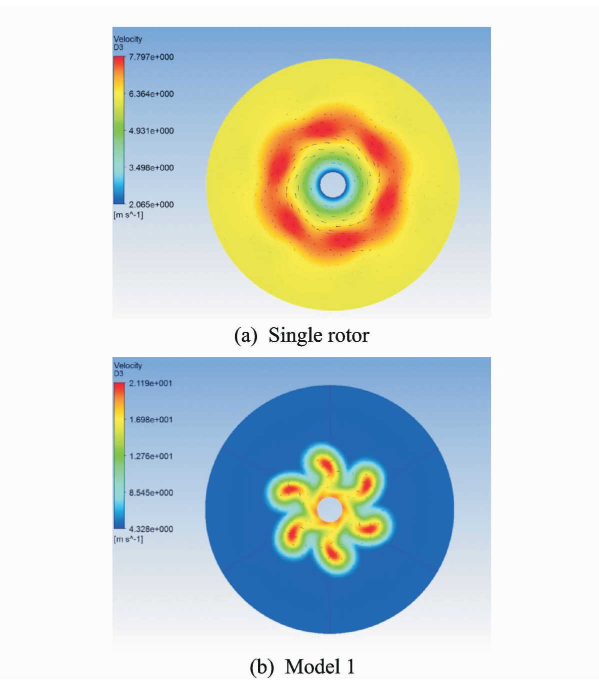
Fig. 13 Velocity contour comparison at 1D behind blades



(a) Single rotor (b) Model 1

Fig. 14 Velocity contour comparison at 2D behind blades

After another diameter far from the blades, the hub vortex and tip vortex of the single rotor develop while the spiral streamlines are occupied the fluid domain (Fig. 14 (a)). Similarly, the flow speeds over the whole domain with the fins are larger than those of the single rotor (Fig. 14 (b)). The area of circumferential velocity of the rotor with the fins is still quite small compared to that of the single rotor. The twist of air flow is limited.



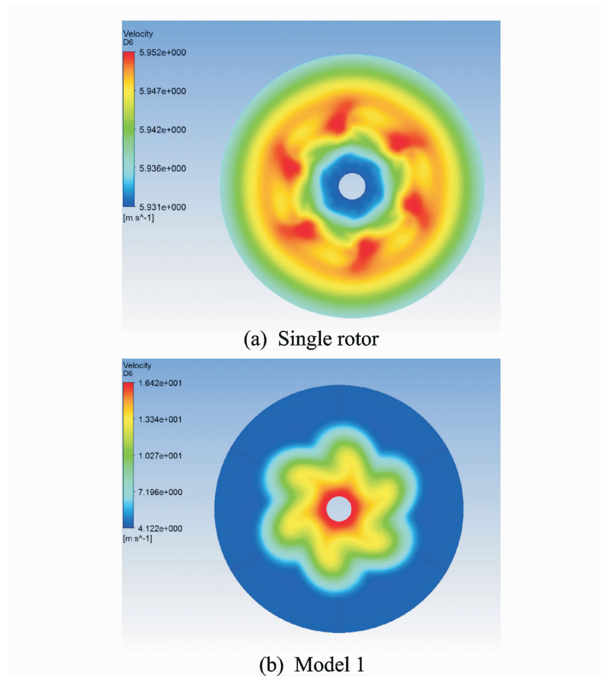
(a) Single rotor

(b) Model 1

Fig. 15 Velocity contour comparison at 3D behind blades

At a distance of 3D behind the blades, the velocity of single rotor become reunion at X direction, but the flows are weakened near the hub area (Fig. 15 (a)). On the contrary, the fins cause the velocity to distribute more evenly, and the velocity strength at hub is much higher than that of the single rotor, which reveals the intensified the air flow at this area(Fig. 15 (b)).

Then, in Fig. 16, near the outlet(6D), the velocities of the airflow become stable and the twist nearly disappear for both single rotor and rotor with the fins. The single rotor has a small velocity around the hub and strong airflow emerges in the tip of the blades. However, the rotor with the fins has condensed airflow with much higher velocity. This is more suitable for generating homogeneous flow.



(a) Single rotor

(b) Model 1

Fig. 16 Velocity contour comparison at 6D behind blades

5 Conclusions

In short, the proper designed fins can effectively help the axial fans for producing homogeneous wind with relatively few efficient cost. The aft fins change the vortex distribution behind the blades and increase the flow uniform and magnitude. The aft fins also increase the flow strength concentrated around the hub. According to the numerical computation, the efficiency can be saved by 11% under the same boundary conditions.

Although the single fan with fins has been discussed in this paper, there are computational effort and experiments needed for analysis on the fan array. The proper quantity of fins and the effect of the diffuser behind the blade will also be investigated in the future work.

Acknowledgements

This research was supported by the State Key Laboratory of Ocean Engineering (No. GKZD010063); and the National Basic Research Program of China ("973" Program) (No. 2014CB046200).

References:

[1] XU B, WANG T. Wind turbine aerodynamic performance prediction based on free-wake/panel model coupled method[J]. Journal of Nanjing University of Aeronauticsand Astronautics, 2011, 43(5):592-597. (in Chinese)

[2] WU J, MENG L, ZHAO Y, et al. Coupled Aerodynamic and hydrodynamic analysis of floating offshore wind turbine using CFD method[J]. Transactions of Nanjing University of Aeronautics and Astronautics, 2016, 33(1):80-87.

[3] COURBOIS A, FLAMAND O, TOULARASTEL J-L, et al. Applying relevant wind generation techniques to the case of floating wind turbines[C]// 6th European and African Wind Engineering Conference. Cambridge, UK: [s. n.], 2013:1-8.

[4] CELIK F, GUENER M. Energy saving device of stator for marine propellers[J]. Ocean Engineering, 2007,34(5/6):850-855.

[5] YU L, DRUNKENBROD M, GREVE M, et al. Numerical and experimental analysis of a ducted propeller designed by a fully automated optimization process under open water condition[J]. China Ocean Engineering. 2015,29(5):733-744.

[6] Ansys CFX 13.0. ANSYS CFX reference guide[M]. USA: ANSYS INC, 2010.

[7] KIM J, PATERSON E, STERN F. RANS simulation of ducted marine propulsor flow including subvisual cavitaion and acoustic modeling[J]. Journal of Fluid Engineering, 2006, 128(4): 799-810.

[8] BERISTAIN I G. Aerodynamic analysis of axial fan unsteady simulations[D]. Spain: University of the Basque Country, 2012.

[9] STEPANOFF A. Centrifugal and axial flow pumps: theory, design, and application[M]. New York: Krieger Publishing Company,1993:504.

[10] HUANG C, GAU C. An optimal design for axial-flow fan blade: Theoretical and experimental studies [J]. Journal of Mechanical Science and Technology, 2012, 26(2):427-436.

[11] YU L, GREVE M, DRUNKENBROD M, et al. Numerical analysis of ducted propeller performance under open water condition[J]. Journal of Marine Science and Technology, 2013, 18(3): 381-394.

Dr. **Yu Long** is a researcher and doctor supervisor at Shanghai Jiao Tong University (SJTU). His recent research interests are hydrodynamic analysis of marine propulsors, turbines and energy saving devices.

Mr. **Huang Chao** is an engineer of SJTU. His research interests cover the marine engine design and fan and ventilation.

Mr. **Zhao Yongsheng** is an engineer of SJTU. His research interests are the offshore floating wind turbine.

Dr. **He Yanping** is a professor and Ph. D. supervisor at SJTU, and his research interests cover the ship design and offshore floating wind turbine.

(Executive Editor: Zhang Bei)