POLISH MARITIME RESEARCH 4 (92) 2016 Vol. 23; pp. 73-83
10.1515/pomr-2016-0072

3-D SIMULATION OF VERTICAL-AXIAL TIDAL CURRENT TURBINE

Zhang Zhiyang,
Ma Yong,
Jiang Jin,
Liu Weixing,
Ma Qingwei,
Shipbuilding Engineering Collage, Harbin Engineering University, Heilongjiang, Harbin, China

ABSTRACT

Vertical-axial tidal current turbine is the key for the energy converter, which has the advantages of simple structure, adaptability to flow and uncomplex convection device. It has become the hot point for research and application recently. At present, the study on the hydrodynamic performance of vertical-axial tidal current turbine is almost on 2-D numerical simulation, without the consideration of 3-D effect. CFD (Computational Fluid Dynamics) method and blade optimal control technique are used to improve accuracy in the prediction of tidal current turbine hydrodynamic performance. Numerical simulation of vertical-axial tidal current turbine is validated. Fixed and variable deflection angle turbine are comparatively studied to analysis the influence of 3-D effect and the character of fluid field and pressure field. The method, put the plate on the end of blade, of reduce the energy loss caused by 3-D effect is proposed. The 3-D CFD numerical model of vertical-axial tidal current turbine hydrodynamic performance in this study may provide theoretical, methodical and technical reference for the optimal design of turbine.

Keywords: Tidal current energy, vertical-axial turbine, hydrodynamic performance, CFD, numerical simulation, 3-D effect

INTRODUCTION

Tidal current energy becomes an alternative of fossil petroleum as it is renewable and greatly reserved across the world [8] [15] [4]. The key equipment to extract tidal current energy is the tidal turbine. Vertical-axial turbine has simple structure and small draft, adapts to various flow directions, rarely faces cavitation problems, therefore it becomes the hot spot in recent research for its many advantages [22] [23]. The purpose of vertical-axial turbine study is to optimize the design, improving the prediction of hydrodynamic performance. Such prediction is based on experimental and theoretical methods. The problem in the experimental prediction is the long period and high cost. Comparatively, theoretical prediction is effective and economic. Hitherto three main theoretical methods are widely used in engineering practice. The first is based on the law of momentum, the second is based on vortex theory, the third is based on CFD for viscous flow [6] [3] [5]. As the rapid development of computer science, CFD methods becomes more and more effective and economic [9] [13] [7] [19] [10] [12]. To accomplish the prediction in a realistic, accurate, and fast way, many researchers have applied dynamical boundary model [2], turbulence model [1] [20] [17], unsteady rotation method, free surface effect, and many other approaches. In 2008 Sun studied the dynamical boundary problem of H-type turbine using slipping mesh method [18]. Yeung et al used STAR-CD to simulate k-ε turbulence model to study cycloid-type vertical-axial turbine [14]. In 2010, a refined UDF control slipping mesh method was applied on cycloid-type turbine by

Luo et al [16, 21, 24]. In 2011, Li studied further in the accuracy and reliability in such CFD simulation, proposing a method to calculate dynamical model under unsteady rotation. This method gave a possibility to simulate freely variable deflection angle turbine[25].

Until now the understanding of the validation of 2-D and 3-D simulation, the mechanism of different performance between 2-D and 3-D simulation, and how to reduce the energy loss caused by 3-D effect is not sufficient. These unknown aspects are key factors in improvement of efficiency. In this study we used CFD method to examine the validation of simulation, establish techniques of vertical-axial turbine simulation, and improve the accuracy of prediction. 2-D and 3-D simulation are validated and comparatively studied. Along with blade optimal control techniques, fixed and variable deflection angle turbine is simulated in 3-D to analysis the influence of 3-D effect and the character of fluid field and pressure field. A method to reduce the energy loss caused by 3-D effect is proposed as a reference for turbine design and optimization

ANALYSIS OF FLUID DYNAMICS

ANALYSIS OF FLUID DYNAMICS

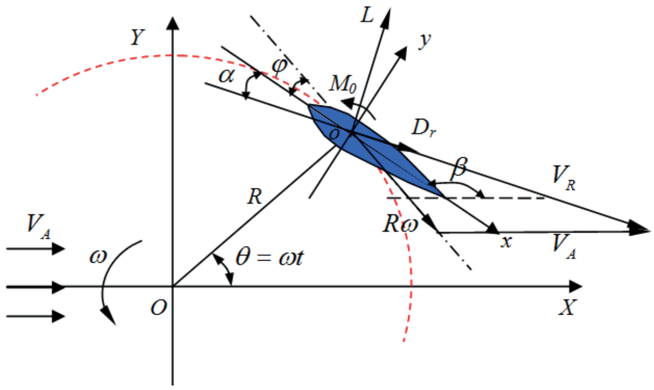

Fig. 1. Coordinate system and forces on the blades

The cross section of the vertical-axial turbine blade is shown as Fig. 1. The blades are uniformly arranged along the circle. The flow velocity is VA, driving the turbine to rotate with angular velocity ω, converting kinematic of the flow to electric power. Here we assume the turbine rotates counter-clockwise and ω is constant. Global coordinate OXYZ is established, the center of the main shaft is located at the original point. The flow is along the X direction. A local coordinate oxyz is attached on any blade. Its self-rotating center is at o the origin. The arc line is parallel to x coordinate, pointing from the end of the blade to the front. θ is location angle, φ is the deflection angle of the blade.

The relative velocity $\vec{V}_R$ of blade and flow satisfies

$$\vec{V}_R = \vec{\omega} \times R + \vec{V}_A \tag{1}$$

$\vec{V}_R$ can be computed by

$$V_R = \sqrt{V_A^{\,2} + (R\omega)^2 + 2R\omega \cdot V_A \sin\theta} \tag{2}$$

The hydrodynamic attack angle α is the angle between VR and x-axis, it is positive when counterclockwise, negative when clockwise.

$$\tan\alpha = \frac{V_A \cos(\theta+\varphi) - R\omega \sin\varphi}{V_A \sin(\theta+\varphi) + R\omega\cos\varphi} \tag{3}$$

As shown in Fig.1, the force acted on a single blade by the flow is resolved into lift force L, damping force D_r, and the momentum M_o to o. We can resolve the force on a blade into to resultant parts, f_x along the arc and f_y perpendicular to the arc, the point of action is o.

$$\vec{f} = (f_x, f_y) = (D_r, L)\begin{pmatrix} -\cos\alpha & \sin\alpha \\ \sin\alpha & \cos\alpha \end{pmatrix} \tag{4}$$

Another way is to resolve the force on normal and tangent directions,

$$\vec{f}' = (f_t, f_n) = (D_r, L)\begin{pmatrix} -\cos(\alpha+\varphi) & \sin(\alpha+\varphi) \\ \sin(\alpha+\varphi) & \cos(\alpha+\varphi) \end{pmatrix} \tag{5}$$

The momentum of a single blade to the turbine is

$$q = f_t R + M_o = R\left[L\sin(\alpha+\varphi) - D_r\cos(\alpha+\varphi)\right] + M_o \tag{6}$$

The above discussion is all about instantaneous quantity of a single blade. When the turbine rotates steadily, the forces on the blade and main shaft are periodic. Assume the number of blades is Z, in a period, the average momentum $\bar{Q}$ and average power $\bar{P}$ are

$$\bar{Q} = \frac{Z}{2\pi}\int_{-\pi/2}^{3\pi/2} q(\theta)d\theta \tag{7}$$

$$\bar{P} = \frac{Z}{2\pi}\int_{-\pi/2}^{3\pi/2} q(\theta)\omega d\theta \tag{8}$$

Here we define the dimensionless coefficients

$$\lambda = \frac{R\omega}{V_A}, \ \mathrm{Re} = \frac{\rho V_A C}{\mu}, \ \sigma = \frac{ZC}{2\pi R}, \ \bar{C} = \frac{C}{R},$$

$$C_P = \frac{P}{0.5\rho V_A^3 Db} = f(\rho, \mu, V_A, w, R, b, Z, C, C_1, \varphi),$$

$$\bar{C}_1 = \frac{C_1}{C}, \ \delta = \frac{b}{C}$$

Where λ is the ratio of blade velocity and flow velocity, R_e is Reynold number, σ is blade denseness, the ratio of sum of arc lengths and orbit circumference, $\overline{C}$ is the ratio of blade arc and diameter, $\overline{C}_1$ is the ratio of the distance from the shaft to the tip and the chord length, δ is the aspect ratio, C_p is the efficient coefficient, μ is the viscosity of the fluid, R is the radius of the turbine, D is its diameter, b is the span of the blade, and P is the power of the turbine.

CONTROL OF DEFLECTION ANGLE

When computing hydrodynamic performance of the turbine, equations 9 and 10 are used to control blade motion[11]. Equations 9 and 10 shows the relation between speed ratio and denseness of a and b. If the location of blade small shaft (0.3 C) and flow velocity (1 m/s) are determined, the values of a and b can be obtained by merely given speed ratio and denseness of the turbine, and therefore obtain the curve of deflection angle. Hence for any turbine model (if the blade type, location of small shaft and flow velocity do not change), the change of deflection angle relating to a specific speed ratio can be obtained.

$$a = \frac{p_1 + p_2 \ln(x) + p_3 \ln^2(x) + p_4 \ln^3(x) + p_5 y + p_6 y^2}{1 + p_7 \ln(x) + p_8 \ln^2(x) + p_9 \ln^3(x) + p_{10} y + p_{11} y^2} \quad (9)$$

$$b = \frac{q_1 + q_2 \ln(x) + q_3 \ln^2(x) + +q_4 y + p_5 y^2 + p_6 y^3}{1 + q_7 \ln(x) + q_8 \ln^2(x) + q_9 \ln^3(x) + q_{10} y} \quad (10)$$

Where x is the speed ratio, y is the denseness, p_1 to p_{11} and q_1 to q_{10} are given in Table 1.

Tab. 1. Coefficients

p_1	2.408E+14	q_1	35.55
P_2	-1.025E+15	q_2	-115.35
P_3	3.520E+15	q_3	72.91
$P4$	-1.360E+15	q_4	131.03
P_5	6.706E+14	q_5	-320.38
P_6	-1.125E+16	q_6	-241.51
P_7	-4.332E+13	q_7	-2.77
P_8	9.200E+13	q_8	0.54
P_9	1.028E+14	q_9	1.50
p_{10}	1.512E+14	q_{10}	1.66
p_{11}	-9.267E+14		

COMPUTING CONFIGURATION AND MESHING

In this study the numerical simulation is mainly based on ANSYS CFX, the entire calculation region is divided into blade region, rotation region, and outflow region. Blade region includes a small region containing the blade, using a cylinder to enclose the blade and making the mesh denser around the blade. Rotation region contains the whole turbine using another cylinder. Outflow region is the entire flow region outside the turbine. Slip mesh is applied to connect the three regions to guarantee the computing accuracy. The meshes of the model are all structural mesh, with refinement around the blade tip. Instantaneous simulation is used, the turbine turns 3° in each time step. The inflow is set uniform and the outflow and surrounding boundary is set open boundary condition. The absolute pressure is 1atm. No-slip condition is used on the blade surface, and turbulence model is chosen as SST model. The mesh of outflow region is shown in Fig.2, blade region in Fig.3, and rotation region in Fig. 4. The meshing details of the blade surroundings are shown in Fig.5, the refinement of blade tip in Fig.6.

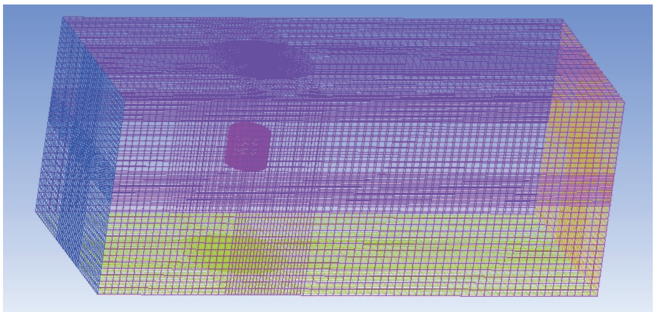

Fig.2. Flow region

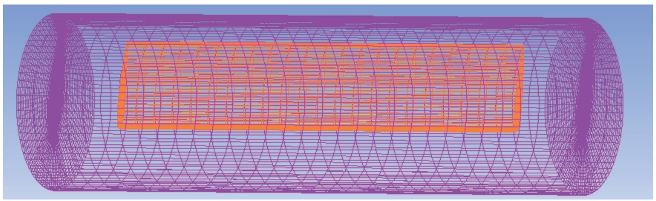

Fig.3. Blade region

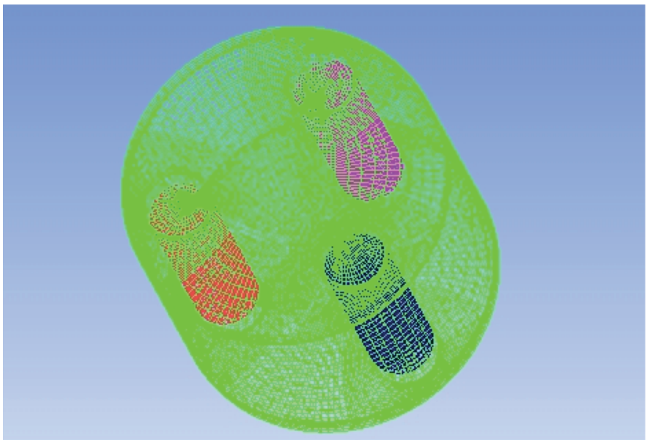

Fig. 4. Rotation region

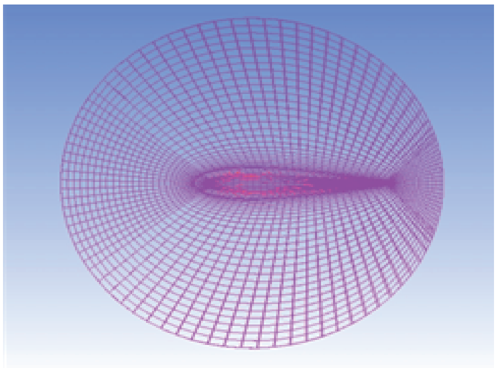

Fig.5. Mesh refinement around the blade

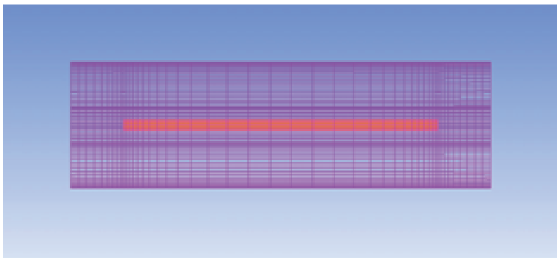

Fig.6. Mesh refinement at blade tip

VALIDATION OF 3-D SIMULATION

Validation of 3-D simulation is demonstrated using NACA0018 blade. The number of blades is 3, the arc length of blades is 0.1524m, arc length to radius ratio is 0.3.As shown in Fig.7, there is difference between the results of numerical method and experiment, where in 2-D simulation the changing trends of these results are the same, but the numerical result is much higher than the experimental result. This is due to the infinite length of the blade in 2-D simulation when 3-D effect is neglected. But when 3-D effect is under concern, the numerical result is close to the experimental result.

SIMULATION OF FIXED AND VARIABLE DEFLECTION ANGLE TURBINE

(1) Fixed deflection angle turbine simulation

The difference between 2-D and 3-D simulation is the span of foil. In 2-D simulation the span of foil is regarded as infinite, while in 3-D simulation it is a determined value. At the top and bottom ends of the blade there is large flow division, many vortices are generated and cause energy lose. Here we choose turbine modes of aspect ratio values 5 and 10 respectively to perform 3-D simulation, then compare with the result of 2-D turbine models. The results are shown in Fig.8.

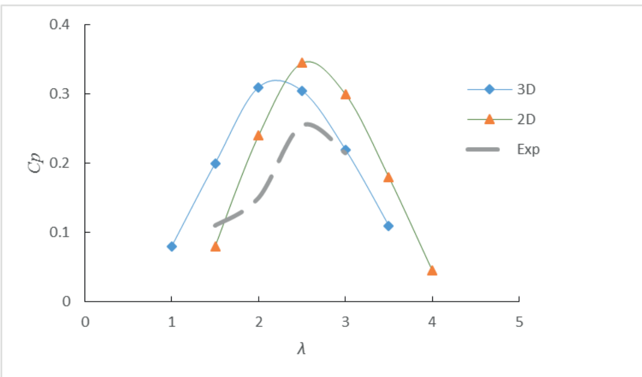

Fig.7. Result of numerical simulation

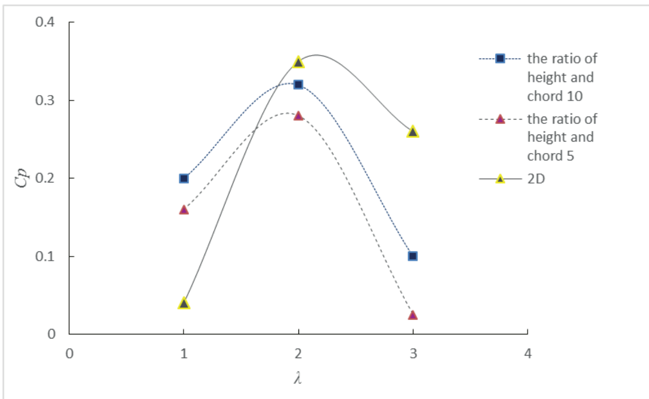

Fig.8. Simulation result of fixed deflection angle turbine

SIMULATION RESULTS AND ANALYSIS

This section investigates the validity of 3-D numerical simulation, influence of 3-D effect and the reduction of 3-D effect, fixed and variable deflection angle turbine are computed, 2-D and 3-D simulation are compared, the case with vortex elimination plate is also simulated.

As shown in Fig.8, when the speed ratio is large, the efficiency is higher in 2-D simulation, which is coincide with the analysis in the last section and interprets that 3-D effect can reduce the turbine efficiency. When the speed ratio is small, 3-D efficiency is much higher than 2-D efficiency, which contradicts the former analysis and needs further discussion. From the comparison between the models with aspect ratios 5 and 10, we find that higher efficiency occurs at larger aspect ratio, indicating that the increase of aspect ratio shall reduce 3-D effect, therefore increasing the aspect ratio can be used to reduce 3-D effect.

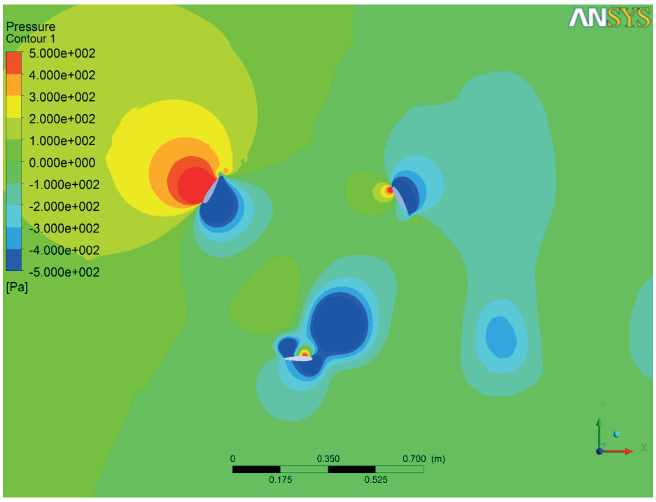

Fig. 9. 2-D simulation pressure nephogram at λ=1

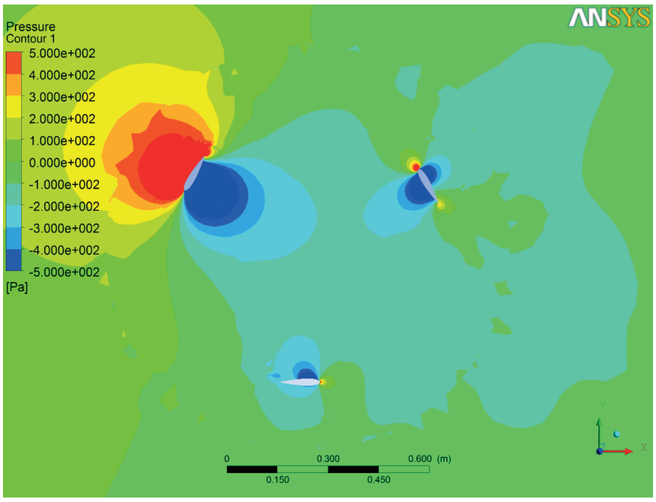

Fig. 12. 3-D simulation pressure nephogram at λ=2

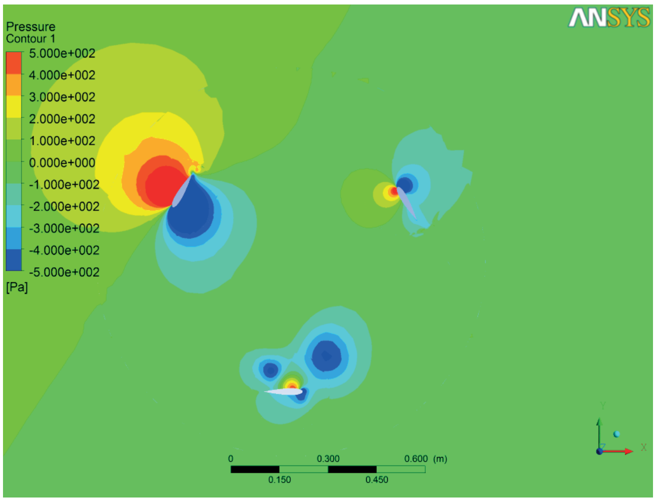

Fig. 10. 3-D simulation pressure nephogram at λ=1

Large error occurs in 3-D simulation when the speed ratio is small. Fig. 9 10 11 12 are pressure nephograms of the flow field in 2-D and 3-D simulations respectively. The turbine is at the same place. We find that the pressure around the blade is lower in 2-D simulation, particularly for the blade of location angle 30°, the whole back of this blade is in low pressure region. But in 3-D simulation only a small part is in low pressure region. Pressure drop on the blade back is often caused by over large attack angle, which leads to flow separation. We can conclude that higher efficiency occurs in 3-D simulation because when flow separation seldom happen, the computing result of lift is larger, therefore the efficiency is larger.

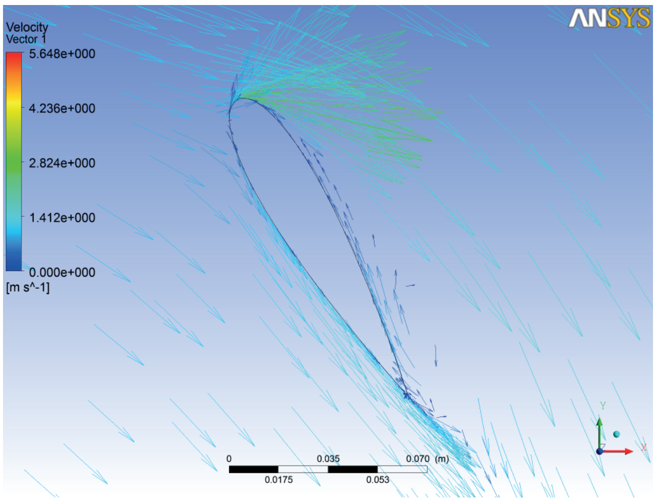

Fig. 13. 2-D velocity vectors at λ=1

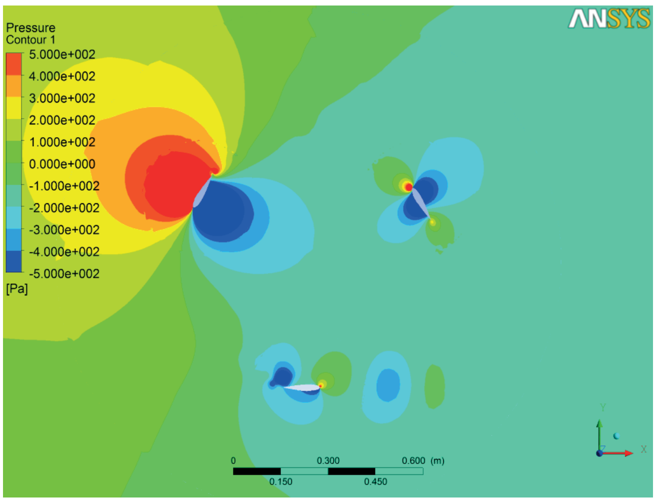

Fig. 11. 2-D simulation pressure nephogram at λ=2

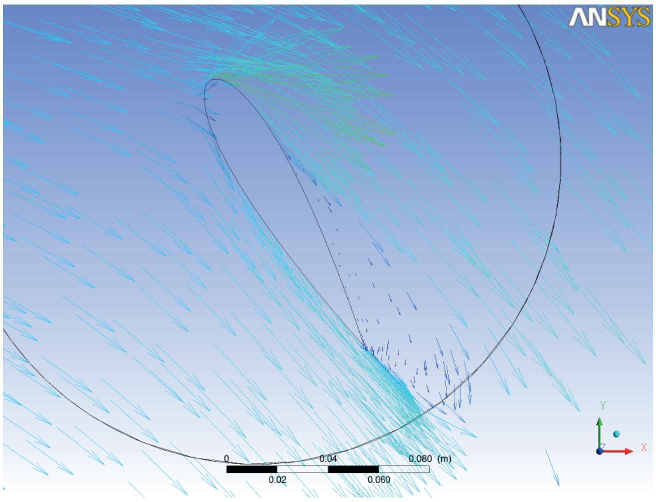

Fig. 14. 3-D velocity vectors at λ=1

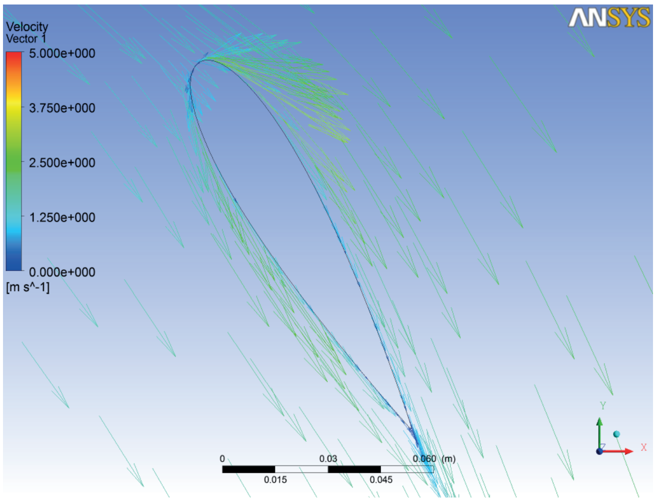

Fig. 15. 2-D velocity vectors at λ=2

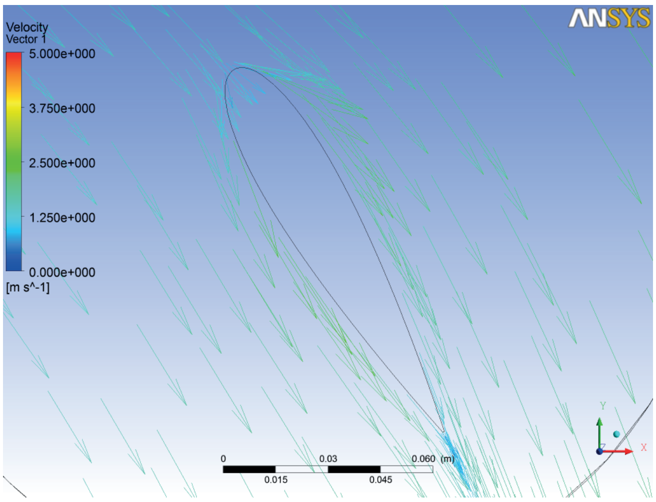

Fig. 16. 3-D velocity vectors at λ=2

When the location of blade is 30°, backflow occurs. In Fig.5.8 backflow occurs only at the root of the blade, this indicates that in 3-D simulation flow separation seldom happens, the same result as in the above section.

Fig.13 to Fig.16 are pressure nephograms and velocity vectors in 2-D and 3-D simulation. No big difference can be found between the two.

The range of attack angle of the blades is wide when the speed ratio is small, the blades often stall under this situation, making the results in 3-D simulation slightly higher, on the other hand, 2-D result is better, therefore 2-D simulation should be employed under small speed ratio. When under medium speed ratio, the range of attack angle becomes narrow, 3-D simulation considers 3-D effect and reflect real physical properties of the model, therefore is more accurate

(2) Variable deflection angle turbine simulation

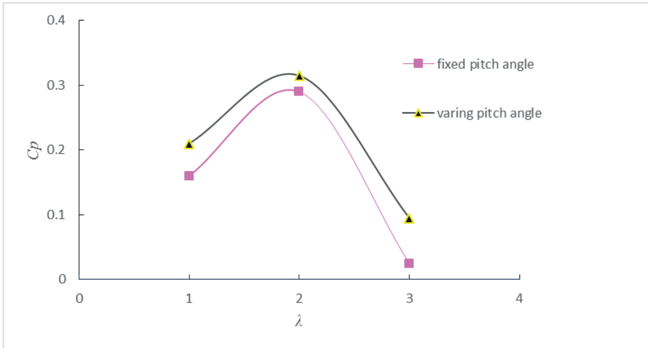

Fig. 17. Comparison of efficiency between fixed and variable deflection angle turbine

The control methods of dynamic mesh in 3-D and 2-D simulations are same, but in 3-D simulation the mesh number is much larger, the computing time is longer. To study the hydrodynamic performance of the variable deflection angle turbine, we compare it with a fixed deflection angle turbine (Fig.17). A model of span of foil equals to 5 is used.

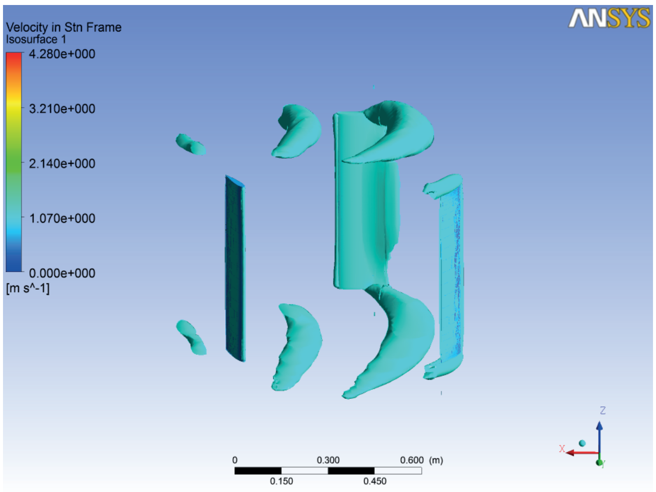

Fig.18 Tip vortices of fixed deflection angle blade at λ=2

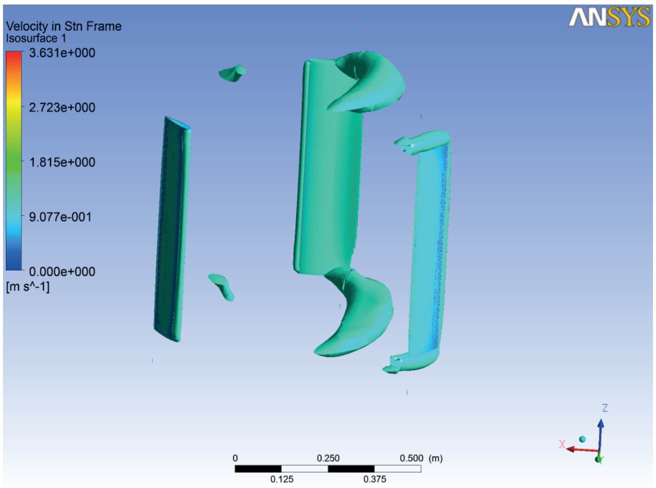

Fig.19. Tip vortices of variable deflection angle blade at λ=2

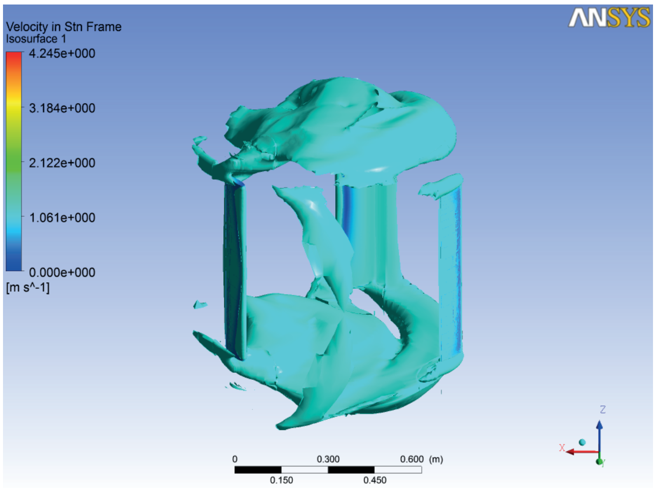

Fig.20. Tip vortices of fixed deflection angle blade at λ=3

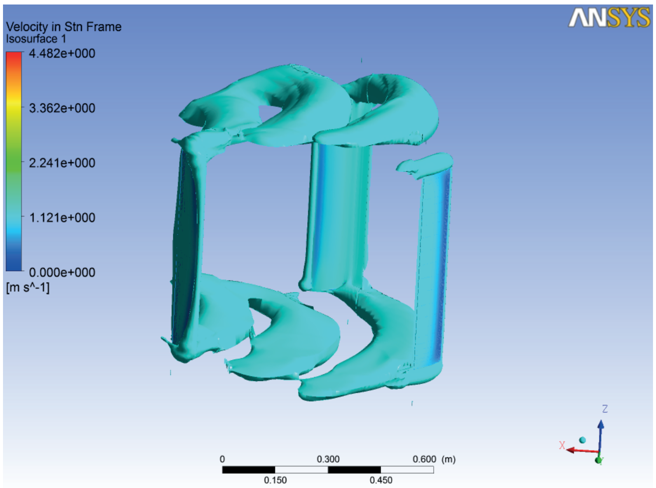

Fig.21. Tip vortices of variable deflection angle blade at λ=3

From Fig.18 to Fig.21, vortices occurs in the wake of the two blade ends, causing energy lose, and this is the greatest difference between 3-D and 2-D simulation, since water flows with different velocities on the top and bottom surfaces of the blade, a pressure difference occurs and push water at the end of blade from high-pressure region to low pressure region, vortices therefore are generated. Else we find that under the same speed ratio, variable deflection angle turbine generate more vortices than its fixed counterpart. This is also the reason that variable deflection angle turbine is more effective. When speed ratio is increased, the vortices become stronger.

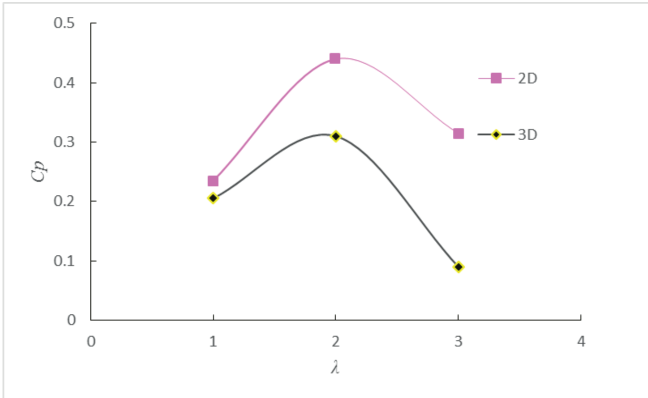

Fig. 22. Comparison of efficiency between 2-D and 3-D variable deflection angle turbine

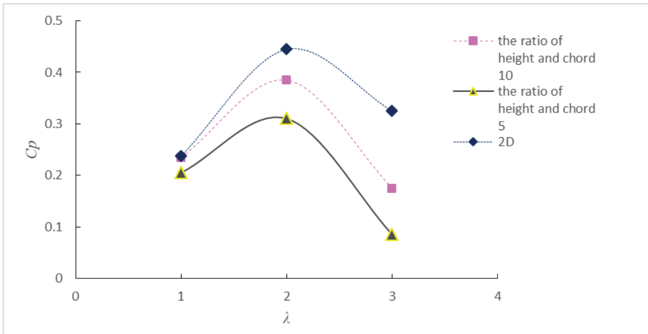

Fig. 23 Influence of aspect ratio

We find that 2-D simulation of variable deflection angle turbine is more effective than that in 3-D simulation, the difference between the two becomes larger as the speed ratio grows. We know that the difference between 2-D and 3-D simulation is the span of foil, and therefore concludes that the influence of aspect ratio becomes larger as the speed ratio becomes larger, this may because as the speed ratio increases, the relative velocity between blade and water becomes larger, more and stronger vortices occurs and take away more energy.

STUDY OF 3-D EFFECT

3-D effect of vertical-axial turbine is mainly caused by aspect ratio. For variable deflection angle turbine with aspect ratio 5 and 10, the compare result of efficiency is shown in Fig.22. From Fig.23 we see that as the aspect ratio increases, the turbine efficiency under each speed ratio raises, but it is lower in 3-D simulation than in 2-D simulation in all cases. Therefore as the aspect ratio increases, 3-D effect becomes weak, and if the aspect ratio increases continuely, the efficiency curve of 3-D simulation shall approach asymptotically to that of 2-D simulation. In the figure we also find that as the speed ratio increases, the difference between the three curves becomes larger and larger, this is because the increase of speed ratio reinforces the influence of 3-D effect. If we intend to reduce 3-D effect we should take the following two methods: (1) make the aspect ratio large, (2) make the speed ratio small.

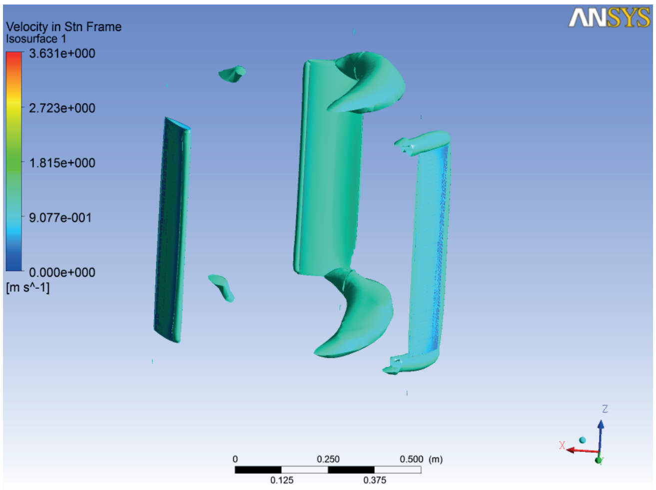

Fig. 24. Vortices at λ=2 and aspect ratio 5

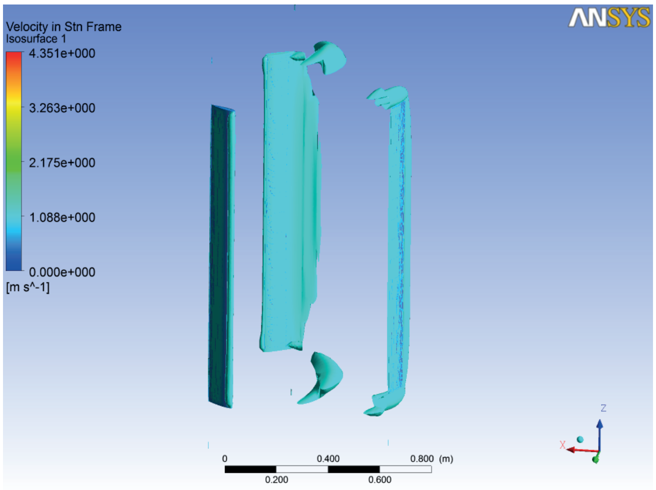

Fig. 25. Vortices at λ=2 and aspect ratio 10

Fig.24 to Fig.27 depict vortices at the blade ends under different aspect ratios. We find that the change of aspect ratio do not influence the vortices much, but when the aspect ratio is large, the total turbine efficiency raises, but the energy lose caused by vortices keep its value, therefore make small effect on the turbine.

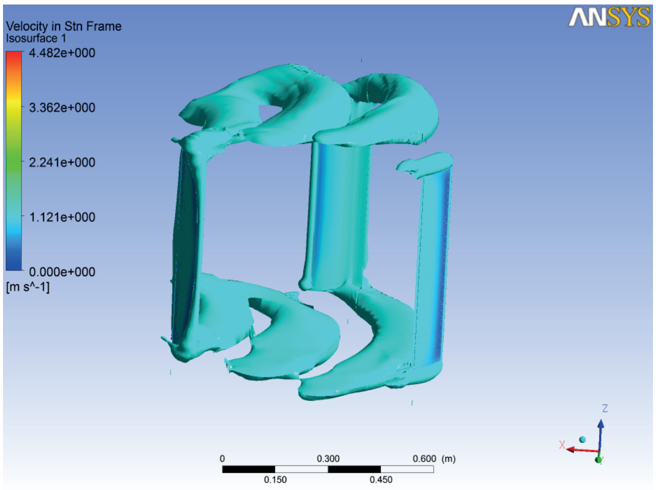

Fig. 26. Vortices at λ=3 and aspect ratio 5

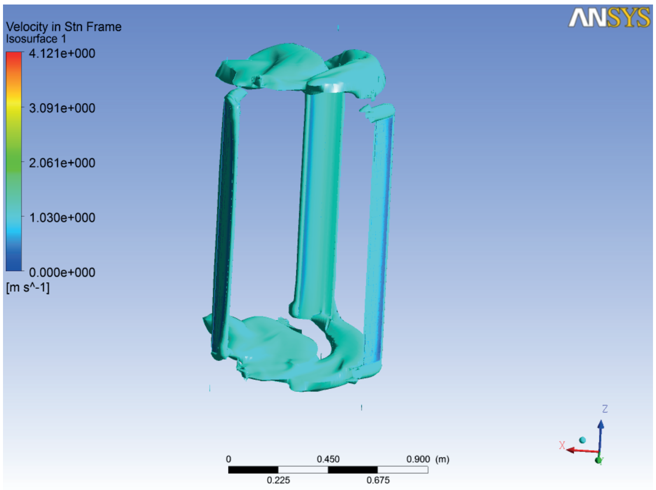

Fig. 27. Vortices at λ=3 and aspect ratio 10

METHOD TO REDUCE ENERGY LOSE CAUSED BY 3-D EFFECT

The vortices at the blade ends cause energy lose. To reduce such lose we can add a plate at the blade ends, which can restrain the generation of vortices. We use CFX software to simulate its performance to test its effect. The model of blade is chosen as NACA0018, the mesh is shown in Fig.28 and Fig.29.

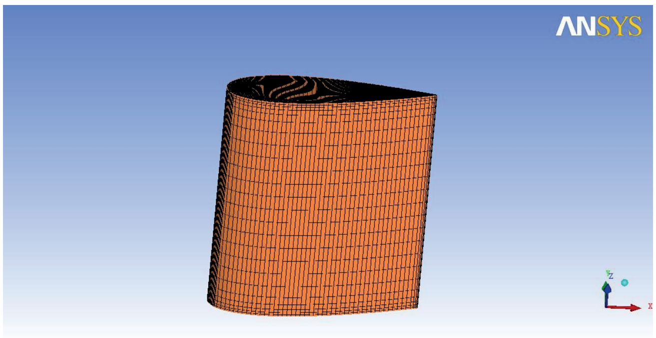

Fig. 28. Blade without plate

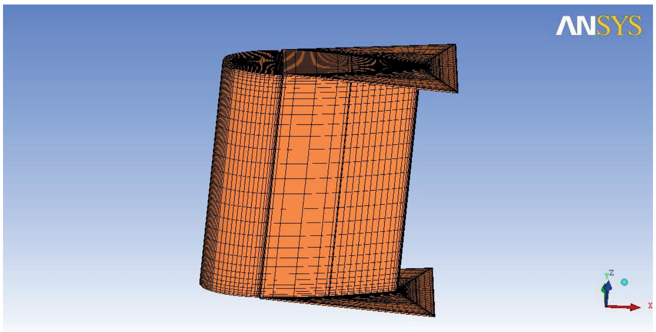

Fig. 29. Blade with plate

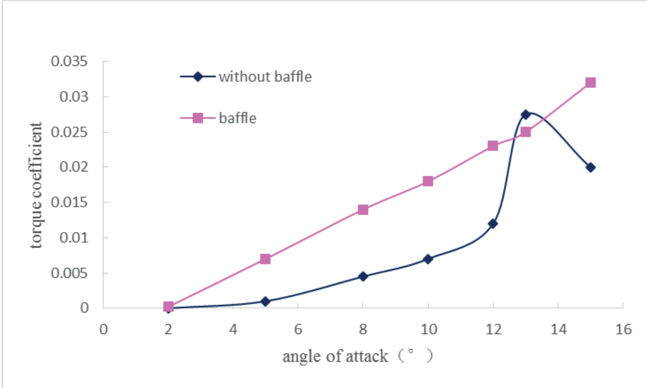

Fig. 32. Comparison of blade momentum

Let the inflow velocity be 3 m/s, using CFX software to simulate the lift coefficient and momentum coefficient of the two kinds of blades with different attack angles. The details are shown in the following figure.

We find from Fig.30 to Fig.32 that when the blades are added plates, its lift coefficient and drag coefficient both have a increase of 10 %, but the drag coefficient is far less than the lift coefficient, therefore the overall performance of the turbine has a considerable reinforcement. As the vertical –axial turbine is lift type turbine, its hydrodynamic performance will be prompted after installing a plate with a 10 % increase of efficiency.

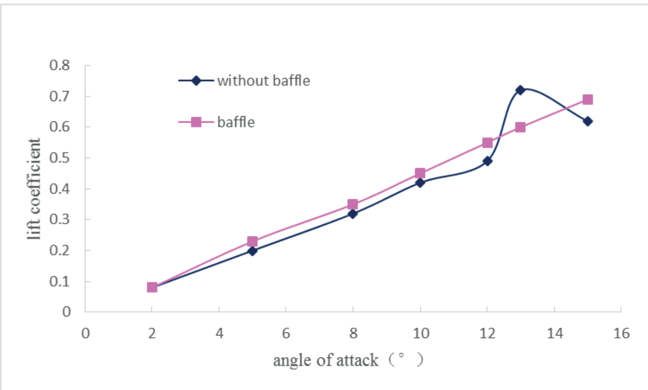

Fig. 30. Comparison of lift coefficient

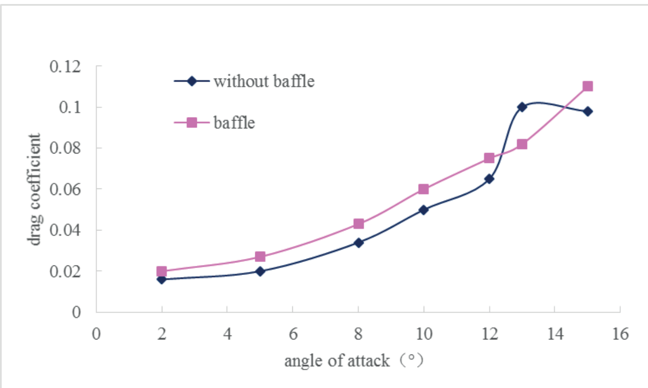

Fig. 31. Comparison of drag coefficient

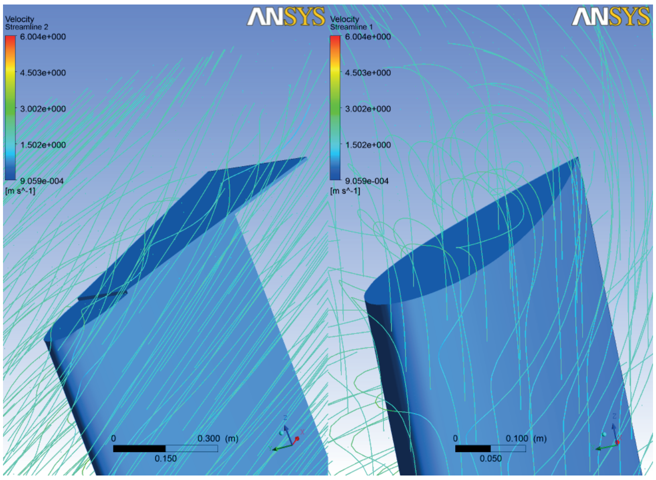

Fig. 33. Comparison of stream line at blade tip with attach angle equals to 15°

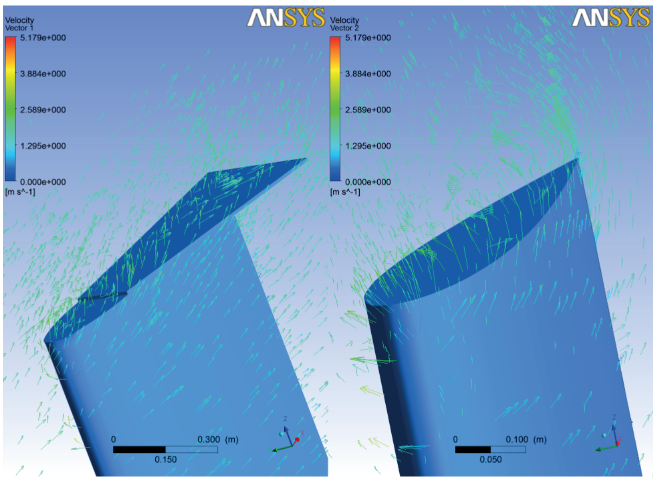

Fig. 34. Comparison of velocity vector at blade tip with attach angle equals to 15°

In Fig.33 (streamline) and Fig.34 (velocity vector), the plates at the ends of the blade can effectively prevent the vortices from being generated, reduce energy lose and enhance its performance.

CONCLUSION

(1) When the speed ratio is large, 3-D simulation is more accurate than 2-D simulation, but under small speed ratio flow separation is much easy to happen, making the results of hydrodynamic performance computation inaccurate.

(2) The energy lose caused by energy lose generated by vortices at the ends of blades is the main reason of difference between 3-D simulation results and 2-D simulation results. Such 3-D effect increases as the speed ratio increases and decreases as the aspect ratio increases. As the aspect ratio becomes very large, 3-D simulation results is close to 2-D results.

(3) Adjusting deflection angle to control the range of attack angle is an effective way to improve the turbine efficiency by 5 % in the method proposed in this paper.

(4) Adding a plate at the ends of blade can effectively reduce the generation of wake vortices and therefore weaken the 3-D effect, the efficieney increases by about 10 %.

ACKNOWLEDGEMENT

This work was supported by National Natural Science Foundation of China (No.51309069, 11572094), the Special Funded of Innovational Talents of Science and Technology in Harbin (No.RC2014QN001008), the China Postdoctoral Science Foundation (No.2014M561334,2015T80330), the Heilongjiang Postdoctoral Science Foundation (No. LBH-Z14060).

REFERENCES

1. F L Ponta, P M Jacovkis. A vortex model for Darrieus turbine using finite element techniques. Renewable Energy, 2001, 24:1-18.

2. G Beer, I Smith, C Duenser. The Boundary Element Method with Programming: For Engineers and Scientists, Springer-Verlag, New York, 2008.

3. Guerri, O., Sakout, A. and Bouhadef, K., Simulations of the Fluid Flow around a rotating Vertical Axis Wind Turbine, Wind Engineering, 2007, 31(3):149-163

4. H F Hassan, A El-Shafie, O A Karim. Tidal current turbines glance at the past and look into future prospects in Malaysia. Renewable and Sustainable Energy Reviews, 2012 16: 5707-17

5. Hamada, K., Smithb, T., Durrani, N., Qind, N. and Howell, R.,Unsteady Flow Simulation and Dynamic Stall around Vertical Axis Wind Turbine Blades, AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, 7-10 January 2008.

6. Horiuchi, K., Ushiyama, I., and Seki, K., Straight Wing Vertical Axis Wind Turbine: A Flow Analysis, Wind Engineering,May 2005, 29(3):243-252

7. H Versteeg, W Malalasekra. An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Prentice-Hall, Upper Saddle River, NJ, 2007.

8. I G Bryden, S J Couch, A Owen et al. Tidal current resource assessment. Proceedings of the Institution of Mechanical Engineering Part A Journal of Power and Energy, 2007 221(2): 125-135

9. J C Tannehill, D A Anderson, R H Pletcher. Computational Fluid Mechanics and Heat Transfer, 2d ed., Taylor and Francis, Bristol, PA, 1997.

10. J D Anderson, Jr. Fundamentals of Aerodynamics, 4th ed., McGraw-Hill, New York, 2007.

11. J. Jiang, Analysis of vertical-axial turbine hydrodynamic performance and application and development of optimal design [D], Harbin: Ph.D. thesis in Harbin Engineering University, 2012.

12. J N Newman, Marine Hydrodynamics, M.I.T. Press, Cambridge, MA, 1977.

13. J Tu, G H Yeoh, C Liu. Computational Fluid Dynamics: A Practical Approach, Butterworth-Heinemann, New York, 2007.

14. Koukina, E., Kanner, S., and Yeung, R. W., "Actuation of Wind-Loading Torque on Vertical Axis Turbines at Model Scale", Proceedings, Oceans'15, Marine Technology Society/IEEE, Paper 141206-002, Genoa, Italy, May18-21,2015

15. L S Blunden, A S Bahaj. Tidal energy resource assessment for tidal stream generators. Proceedings of the Institution of Mechanical Engineering Part A Journal of Power and Energy, 2007 221(2): 137-146

16. L Zhang, Q J Luo, R G Han, Deflection angle optimization of vertical axis tidal current turbine. Journal of Harbin Institute of Technology, 2011 43 (sup1): pp. 281-285.

17. M.S.U.K. Fernando and V.J. Modi. A Numerical Analysis of the Unsteady Flow Past a Savonius Wind Turbine. J. Wind Engineering & Industrial Aerodynamics. October 1989,32(3),303-327P

18. Sun Ke, Numerical simulation of H type vertical-axial turbine and deflector, HEU Ph.D. thesis, 2008: pp. 44-51

19. T Cebeci. Computational Fluid Dynamics for engineers. Springer-Verlag, New York, 2005.

20. Y Li. Development of a Procedure for Predicting the Power Output from a Tidal Current Turbine Farm, PhD thesis, University of British Columbia, Vancouver, B.C.,Canada, 2008

21. M Tomera. Dynamic positioning system for a ship on harbour manoeuvringwith different observers. Experimental results. Polish Maritime Research. SEP 2014, 21(3): 13-24.

22. Y Li, S M Calisal. Modeling oftwin-turbinesystemswithve rticalaxistidalcurrentturbines: Polish Maritime Research, 2010 37: 627-637

23. Y Li, S M Calisal. Modeling oftwin-turbinesystemswithver ticalaxistidalcurrentturbines: Part II—torque flauctuation. Ocean Engineering, 2011 38: 550-558

24. V V Klemas, Coastal and Environmental Remote Sensing from Unmanned Aerial Vehicles: An Overview. Journal of Coastal Research. SEP 2015, 31(5): 1260-1267.

25. Z C Li, L Zhang, K Sun, X W Zhang, Numerical simulation of vertical axis tidal current turbine. ACTA ENERGY SOLARS SINICA, 2011 32 (9): pp. 1321-1326.

CONTACT WITH THE AUTHOR

Ma Yong

Shipbuilding Engineering Collage
Harbin Engineering University
Heilongjiang, Harbin, 150001

CHINA