Proceedings of the 3rd International Conference of Energy Harvesting, Storage, and Transfer (EHST'19)
Ottawa, Canada – June, 2019
Paper No. EHST 141
DOI: 10.11159/ehst19.141

Influence of Mesh Resolution of Boundary Layer on Numerical Simulation of Full-scale Tidal Turbine

Xiancai Si[1,2], Peng Yuan[1,2], Shujie Wang[1,2], Junzhe Tan[1,2], Xiaodong Liu[1]
[1]College of Engineering, Ocean University of China
Qingdao, 266100, China
sixiancai@ouc.edu.cn
[2]Ocean Engineering Key Lab of Qingdao
Qingdao, 266100, China

***Abstract*—**Boundary layer mesh is very sensitive to numerical simulation results. However, in the simulation of high Reynolds number, the boundary layer requires more computational cost and is complicated to implement. Therefore, under the premise of limited computational cost, how to reasonably set the boundary layer mesh and predict its calculation error is very necessary. In this paper, a full-scale horizontal axis tidal turbine, used as the research object, adopted by CFD software for RANS computations of high Reynolds number turbulent flow. Then, the fluctuation range of numerical simulation results under different mesh resolutions of blade boundary layer was investigated. Influence of mesh resolution of the boundary layer on the simulation results of the turbine's torque was significant, which could lead to fluctuation more than 20% of the simulated value; the influence on the axial force was small, which was no more than 8% of the fluctuation results. In addition, simulation results showed mesh resolution (y+) in the boundary layer should not be greater than 300, otherwise the simulation value were similar to the results without the boundary layer at all, but it would increase the redundancy calculation.

***Keywords*:** Boundary layer, CFD, Structured grid, Tidal turbine.

1. Introduction

In fluid mechanics, boundary layer theory scientifically unifies the interaction of fluids and solids[1]. However, in the study of practical engineering problems, especially in the numerical simulation of high Reynolds number, the implementation of the boundary layer is difficult and cost more computing resources. Therefore, it is very meaningful to rationally set the mesh resolution of boundary layer to save computational resources while ensuring certain calculation accuracy, especially in the fluid medium at high flow rate.

In the paper, a full-scale horizontal axis tidal turbine, developed by Ocean University of China[2], was used as the study target. Then, a numerical model of a full-scale horizontal axis turbine with its ambient flow fields was constructed using computational fluid dynamics(CFD). Based on the numerical model, the simulation results under different mesh resolutions of blade boundary layer were studied. The study can provides a reference for numerical simulation of full-scale tidal turbine, which ensures the mesh resolution of boundary layer be set more reasonably under the condition of limited comuting resources.

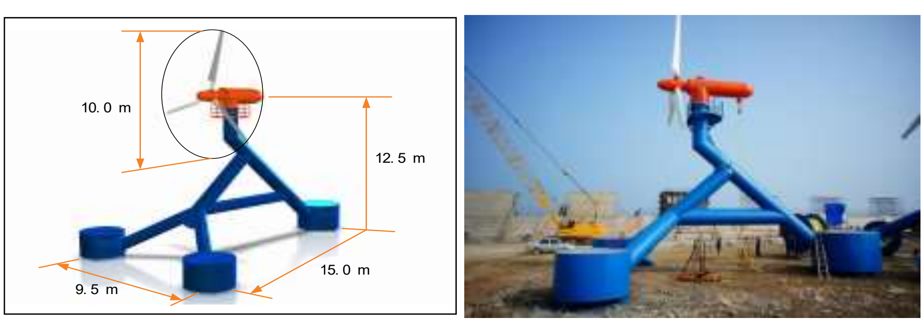


Fig. 1: Specification of the "Haiyuan I" horizontal axis tidal current turbine.

2. Methodology

2.1. Research object and mesh scheme

The investigated object is the “Haiyuan I” horizontal axis tidal turbine, as shown in Fig.1, which is a gravity-based structure developed by Ocean University of China . The diameter of the turbine blade is 10.24 m, and the distance between the centre of the hub and the bottom of the support is 12.5 m. The rated flow velocity is 1.7 m/s. The turbine yields maximum power efficiency at a tip speed ratio of 7, which means the turbine will take 2.7 s for one rotation cycle at a rotational speed of 2.3 rad/s.

As the blade and the rotor of the turbine are the objects of concern in this study, other auxiliary and supporting components of the turbine are neglected. This simplification reduces the number of grids generated in the final computation domain, thus speeding up the computation and improving the efficiency. This practice is very common in wind turbine research [3] and is also applied in tidal turbine research [4]. Existing experimental results confirm that this simplification affects the flow structure and behaviour of the downstream wake to a certain extent, and the effect on the force and power of the blade is almost negligible [5].

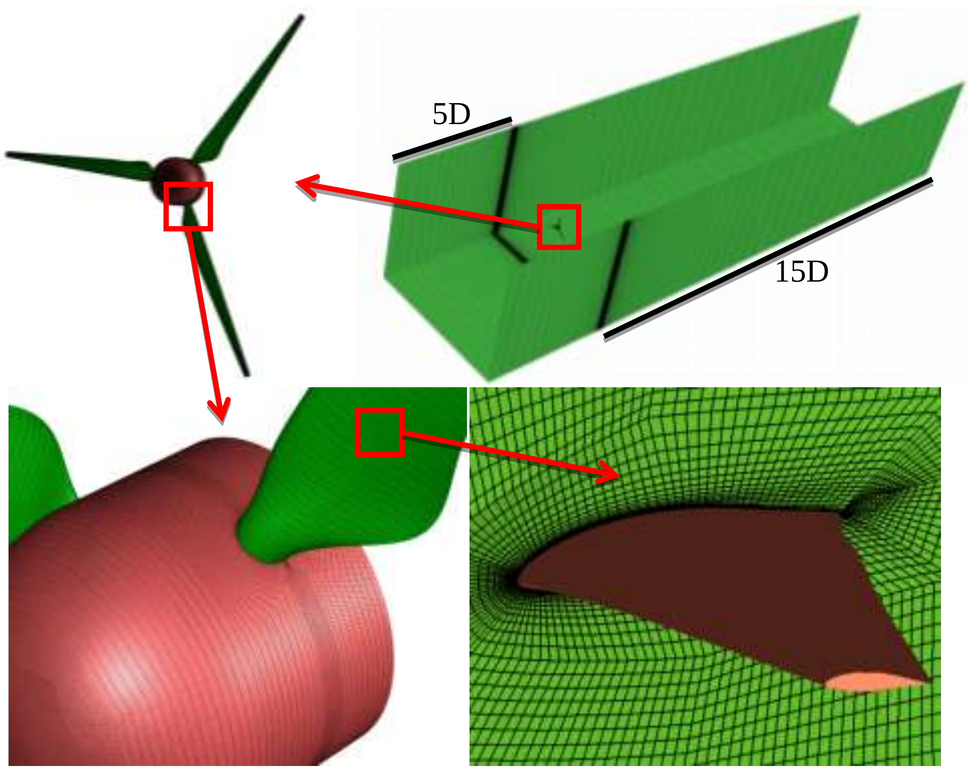


Fig. 2: Mesh scheme.

In this study, ICEM was used to build the computational domain, as shown in Fig.2, which was divided into two parts: a rotating domain and a fluid domain, and the interface was used to transfer data between the two domains.

In the fluid domain, the total length of the computational domain was 20D (D represents the diameter of the turbine), and it had three parts in the direction of the incoming flow, namely the upstream part of the turbine (its length was 5D), the main part of the turbine, and the downstream of the turbine (its length was 15D). In addition, to minimize the blocking effect of the turbine for the upstream flow, the width of the computational domain was set to 6 times the diameter of the turbine.

In the rotating domain, the turbine segment consists of a rotating cylinder domain containing the rotor and a rectangular fluid domain that cuts the cylinder. The cylinder rotating domain contains three blades and a hub. The boundary layer mesh at the interface between the blade and the fluid was refined by an O-type block to facilitate the setting of the boundary layer, and a total of 20 layers of mesh were set in this part. In addition, considering the limited pages of the paper, mesh independent analysis is no longer stated in detail.

2.2. Boundary layer theory

In the flow with a high Reynolds number, flow velocity is closely affected by the viscosity of the surface of the object. This layer of very thin fluid, which cannot be neglected by friction shear stress, is the boundary layer. In numerical simulation, there are usually two ways to deal with near-wall simulations: (1) using the wall function method; (2) refining mesh, and using the wall model method. The selection of these two methods can be expressed by y+.According to the boundary layer theory, in the near wall region, the spatial velocity gradients are large. Hence a fine grid resolution is required to accurately resolve them. Considering that the current research target was a full-scale turbine, it was difficult to investigate the simulation results in the viscous sublayer. So, the size of the boundary layer mesh selected for this study was y+ from 80 to 480, the thickness of first layer mesh was 0.2mm-1.2mm under turbine rated conditions.

2.3. Solver and turbulence model

Calculation was carried out using the universal CFD solver FLUENT. A pressure-based solver was utilized for the entire simulation with transient mode, and the blades rotation speed was set to1° at the start of each time step.

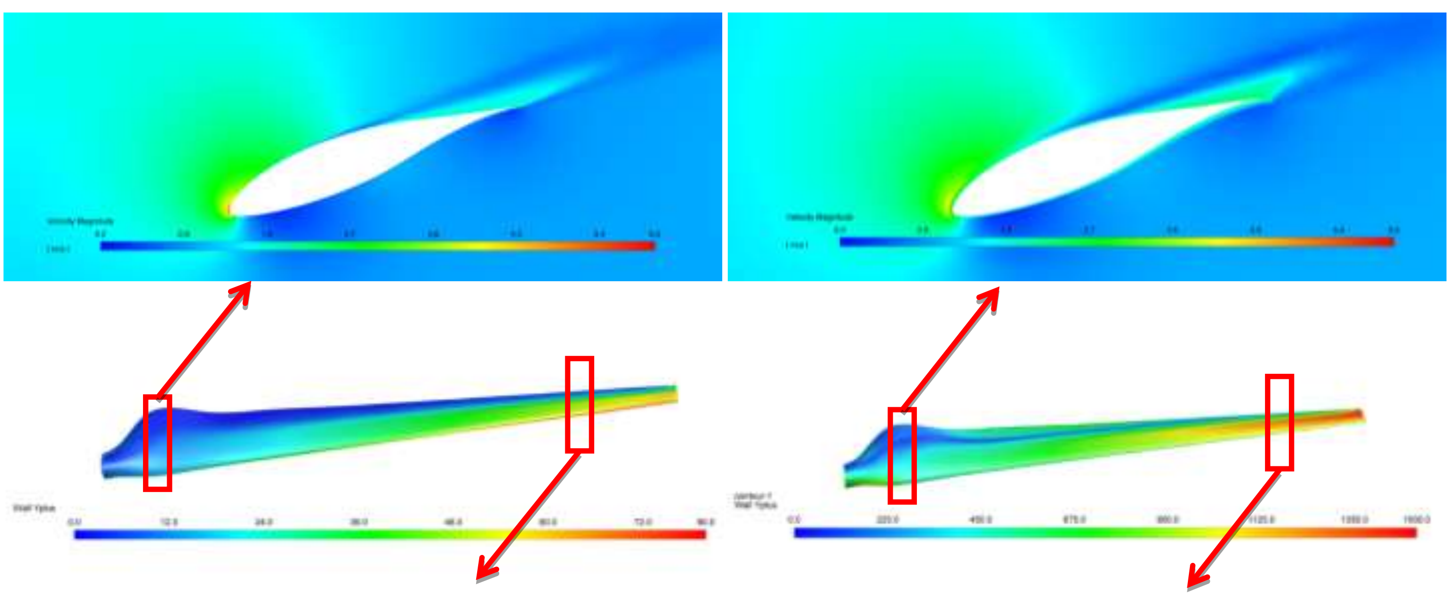
The k-ω SST turbulence model was adopted for numerical simulation, which was developed by combining the stability advantages of the standard k-ε and k-ω turbulence models [6]. Near the wall area, the wall function was implemented, and the k-ω model was applied to calculate the area far from the wall. This ensures that the model is very robust in dealing with a complex flow boundary, which is widely employed in many similar research fields [7].

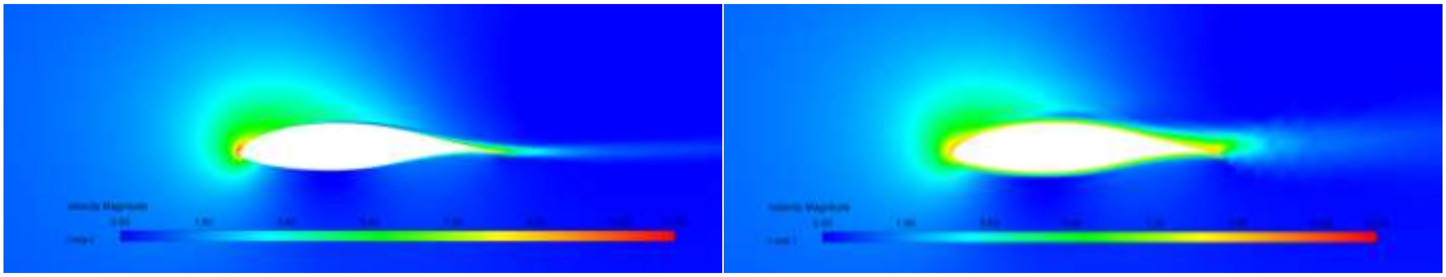
3. Discussion

According to the study results, the boundary layer had a great influence on the predicted value of the blade, specially at the leading and trailing edges of the blade, as shown in Fig.3. At the leading edge, simulated flow velocity adopting boundary layer was significantly higher than without boundary layer. At the trailing edge, it was difficult to accurately calculate the boundary layer separation and wake characteristics without the boundary layer.

Comparing the mesh size of different boundary layers, the smaller the mesh size of the boundary layer, the longer the time required for the model to stabilize, especially in the torque of the blade, as shown in Fig.4.

Compared to the case where the boundary layer is not considered, the influence of mesh size of the boundary layer on the simulation results of the turbine’s torque was significant, which could lead to more than 20% of the simulated fluctuations; the influence on the axial force was small, which was no more than 8% in the fluctuation of numerical results, as shown in Fig.5. It is preliminarily guessed that the boundary layer might have a greater impact on the simulation of the machine of lift driver, but less on the machine of drag driver.





(a) Considering boundary layer (b) Without considering boundary layer

Fig. 3: Simulation in blade section. (The turbine blade radius is 5.12m, and the screenshot positions are 1m and 5m in the radial direction of the blade).

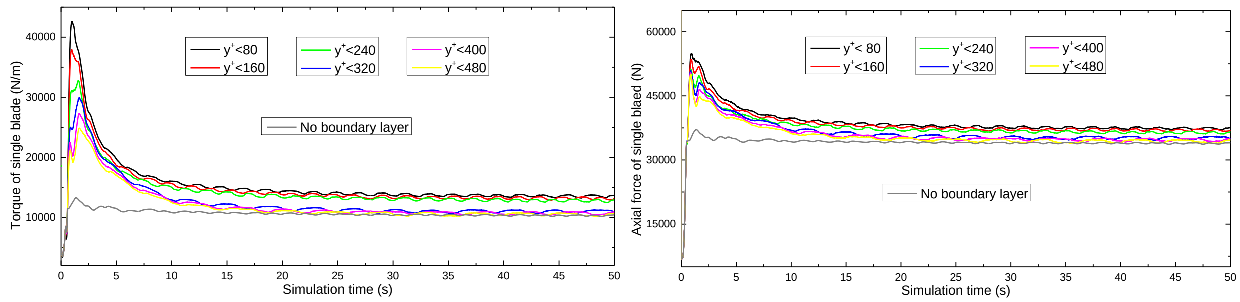


Fig. 4: Model stabilization time.

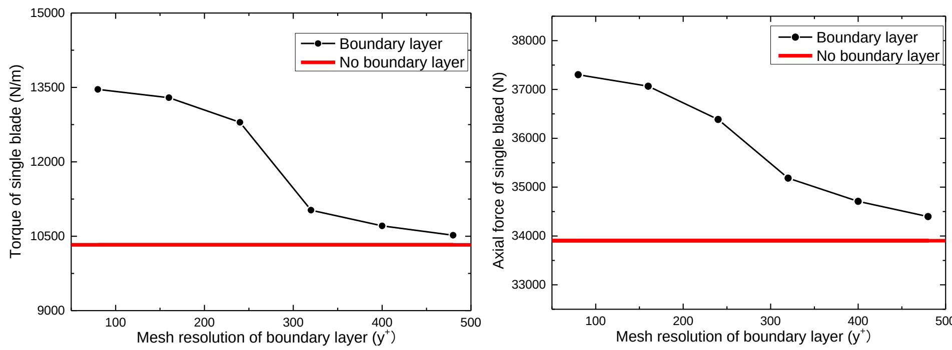


Fig. 5: Comparison in different mesh resolution.

In addition, while the mesh size of the boundary layer (y+) was between 200 and 300, the simulation were sensitive, the fluctuation of which might be very large. At less than 200, the simulation tend to be stable and there was a tendency to converge; at more than 300, the simulation were not much different from the simulation without considering the boundary layer. So, the mesh resolution in the boundary layer should not be greater than 300, otherwise the simulation were similar to the results of not considering the boundary layer at all, but it would increase the redundancy calculation.

4. Conclusion

In the paper, a full-scale horizontal axis tidal turbine, developed by Ocean University of China, was adopted as the research target. Based on universal CFD solver, a turbine model was built to investigate the fluctuation range of simulation results under different mesh resolutions of blade boundary layer. The research results shown:

The mesh resolution in boundary layer could lead to more than 20% of the simulated fluctuations in turbine's torque; and no more than 8% of the simulated fluctuations in the axial force.

In the boundary layer, the mesh size (y+) in first layer should not be greater than 300, otherwise the actual effect would not be obtained.

Acknowledgements

The study is supported by the Special Projects of Renewable Energy and Hydrogen Energy Technology for National Key R&D Program (No. SQ2018YFB1501903) and Shandong Provincial Natural Science Key Basic Program (Grant No.:ZR2017ZA0202)

References

[1] F. M. White. *Fluid Mechanics.* 5th edition, Boston: McGraw-Hill Higher Education, 2008, page 467.

[2] S. Wang, P. Yuan, J. Tan, "Research and tests on hydrodynamic performance of a seabed seated horizontal axis tidal turbine," in *Proceedings MTS/IEEE Oceans 2015*, Genoa.

[3] F. Zahle, N. N. Sørensen, J. Johansen, "Wind turbine rotor-tower interaction using an incompressible overset grid method," *Wind Energy*, vol. 12, pp. 594-619, 2009.

[4] T. P. Lloyd, S. R. Turnock, V. F. Humphrey, "Assessing the influence of inflow turbulence on noise and performance of a tidal turbine using large eddy simulations," *Renew Energy*, vol. 71, pp. 742–54, 2014.

[5] S. Kang, I. Borazjani, J. A. Colby, F. Sotiropoulos, "Numerical simulation of 3D flow past a real-life marine hydrokinetic turbine," *Adv Water Resource*, vol. 39, pp. 33-43, 2012.

[6] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA J*., vol. 32, pp. 1598-605, 1994.

[7] P.C. Rocha, H. B. Rocha, F. M. Carneiro, M. V. da Silva, A. V. Bueno, "k-ω SST (shear stress transport) turbulence model calibration: A case study on a small scale horizontal axis wind turbine," *Energy*, vol. 65, pp. 412–8, 2014.