NUMERICAL VALIDATION OF WAKE INTERACTION BETWEEN TWO OFFSET MODEL WIND TURBINES BASED ON ACTUATOR LINE MODEL

XINZE DUAN[1], PING CHENG[2], DECHENG WAN[*]
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai 200240
**Corresponding Author: dcwan@sjtu.edu.cn*

Keywords: *Wind farm; Actuator line model; Wake interaction; Blind Test 3; Tip speed ratio*

In large wind farms, some wind turbines are disturbed by the wake of other turbines. The wake interaction phenomenon among wind turbines has a great influence on aerodynamic power output, wind speed deficit turbulence stress and wake vortex structure, which indicates that more attention should be placed on the wake interaction for the optimal arrangement of wind farm. In this paper, the actuator line model combined with CFD technique will be applied to study the wake interaction between two offset wind turbines. The results obtained from the present simulations are compared to the data from the experiment “Blind Test 3” and other simulation models. From the comparison, the results from the present study show a good agreement with the experimental results especially for the aerodynamic loads prediction, whose error is not over 3% for the upstream wind turbine. Although the simulations for two offset wind turbines model exist some difference in wake prediction compared to the “Blind Test 3”, the overall change trend is consistent with the experiment. Based on the results, the actuator line model can simulation the aerodynamic loads of the two offset model wind turbines and basic features of the wake, including the distribution characteristics of the mean wake velocity and mean turbulent stress.

Introduction

The wake interaction among wind turbines will lead to the decreased inflow wind velocity and increased turbulence intensity. Decreased total production of power and increased levels of fatigue loads are imposed on the turbines in the wind farm due to the phenomenon of wake interaction[1]. Therefore, how to accurately simulate the wake interaction among multiple wind turbines is becoming the key problem to improve the efficiency of the wind farm.

In order to study the complex phenomenon of wake interaction in wind farms, some scholars attempt to model the flow field in wind farms are based on CFD approach. Frandsen[2] pointed out that the advantage of using CFD approach is that they, besides avoiding the scale effect in the model experiment and handling the large Reynolds number problem well, also provide necessary information about the wake characteristics. A mixed method named actuator line model (ALM) combined with CFD technique was developed by Sørensen and Shen[3] is an extensively used method to model the rotor as force field. Some researchers have done lots of work about the wind farm simulation using actuator line model. Churchfield[4][5] have done a large-eddy simulation of the Lillgrund wind plant which contains 48 multi-megawatt turbines, and turbines were modeled using actuator line representation. Ai Yong[6] presented numerical simulations of the effects of inter-turbines spacing changed from three to nine times of rotor diameter on aerodynamics for wind farms contained two NERL 5MW baseline wind turbines in tandem layout.

In this present study, a numerical validation of CFD combined with actuator line methods for modeling the rotors is conducted. The results obtained from the present simulations are compared to the data from the experiment “Blind Test 3” and other simulation models including the aerodynamic loads, the mean wake velocity and mean turbulent stress.

Numerical modelling

The actuator line model (ALM) was firstly developed by Sørensen and Shen[7]. The rotating blades are virtualized into span wise sections of constant airfoil, chord and twist and the forces are distributed over them. Hence, there is not requirement to build the actual blades model. The local velocity U_{rel} relative to the rotating blade is calculated as:

$$|U_{rel}| = \sqrt{(U_{in})^2 + (\Omega r - U_{rot})^2} \tag{1}$$

where U_{in} represents the inflow wind velocity, Ω is the angular velocity of the rotor. The inflow angle is determined as:

$$\phi = tan^{-1}\left(\frac{U_{in}}{\Omega r - U_{rot}}\right) \tag{2}$$

The local angle of attack is given by $\alpha = \phi - \theta_t$, where the θ_t is the local twist angle. Having determined the angle of attack and relative velocity, the lift and drag force per span wise length can be got:

$$f = (L, D) = \frac{\rho |U_{rel}|^2 c N_b}{2rd\theta dz}(C_L e_L + C_D e_D) \tag{3}$$

where c is the chord length; N_b is the number of blades; C_L and C_D are the lift and drag coefficient, respectively; The aerodynamic blade force that cannot be directly applied to the flow field, it need to be distributed smoothly on the flow field volume in order to avoid singular behaviour. In practice, a 3D Gaussian function is made to smooth the force over the blade by taking the convolution of the force with a regularization kernel.

$$f_\varepsilon = f \otimes \eta_\varepsilon \tag{4}$$

$$\eta_\varepsilon(d) = \frac{1}{\varepsilon^3 \pi^{3/2}} exp\left[-\left(\frac{d}{\varepsilon}\right)^2\right] \tag{5}$$

where d is distance between cell-centered grid points and the actuator line point, and ε is parameter that serves to control the width of Gaussian and to adjust the concentration of the regularized loads.

The *k-ω* SST turbulence model is applied to solve the RANS equation. And the governing equations can be written as:

$$\nabla \cdot U = 0 \tag{6}$$

$$\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial t}\left(U_i U_j\right) = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial U_i}{\partial x_j} - \overline{\acute{u}_i \acute{u}_j}\right) + \frac{1}{\rho} f_\sigma \tag{7}$$

where U is the velocity of flow; ρ is the density of the fluid; p is the pressure;ν is the kinematic viscosity, f_σ denotes the body forces, which represent the loading on the rotating blades. The body force acting on the blades are determined using a blade element method combined with tabulated two-dimensional airfoil characteristics.

Experiment Description and Simulation Setup

The Blind test 3 was organized by Norcowe and Nowitech in Bergen, 10 and 11 December, 2013. The wake development behind two model wind turbines that have been extensively tested in the large close-loop wind tunnel facility at NTNU. The two turbines were arranged offset so that downstream wind turbine was affected by the partial impingement of the wake developed by the upstream turbine. Sketches of the layout are given in Fig. 1 shows a picture of the turbines mounted in the tunnel. The wind tunnel has a rectangle test section, whose dimensions at the inlet are W=2.72 m and H=1.80m (W means width and H means height). In addition, a large scale bi-planar grid was mounted at the entrance to the test section in order to make the conditions more similar to the atmospheric conditions. The reference velocity was set to U_∞ = 10 m/s. At this velocity, the turbulence intensity is TI =10% at the inlet. More detail about the "Blind Test 3" can be found in reference[8][9].

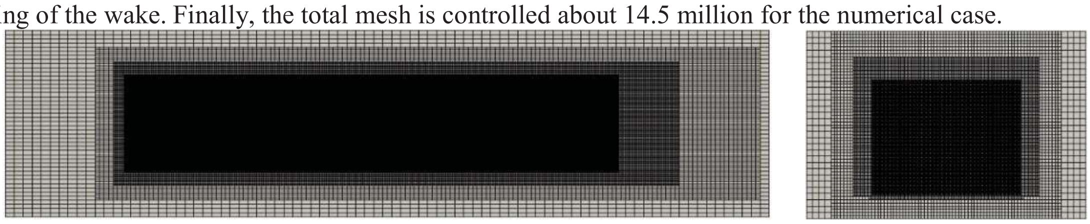
Figure 1: wind tunnel

Furthermore, the two wind turbines have the same blade geometry, but slightly different hub size, thus leading to different rotor diameter. The diameters of upstream and downstream wind turbine are 0.944 m and 0.894 m, respectively. Table 1 gives some specification of the two wind turbines used in this present study. Where WT1 represents the upstream wind turbine and WT2 is the downstream wind turbine.

Table 1. Parameters of two wind turbines

Items	WT1	WT2
Airfoil	S826	S826
Rotor Diameter	0.944 m	0.894 m
Nacelle Diameter	0.13 m	0.08 m
Height of tower	0.817 m	0.817 m
Pitch Angle	0	0
Tip Speed Ratio	6	4.75

In order to compared to "Blind test 3", the computation of the numerical study is stayed as the same dimensions with the wind tunnel in NTNU. Meanwhile, the wind turbines maintain the same layout. Furthermore, the mesh has been densified in order to fully resolve the strong gradients in the vicinity of the actuator lines and carefully observe significant meandering of the wake. Finally, the total mesh is controlled about 14.5 million for the numerical case.



(a) Grid in cross section (b) Grid in lengthwise section

Fig. 2 The grid in lengthwise section and cross section

In this study, the uniform free-stream flow condition whose values is 10 m/s at the reference height of hub is applied to the inlet. A relative pressure of 0 Pa based on the atmospheric pressure is chosen for outlet boundary. Considering the blockage effects, the wall condition is applied to the top, bottom and sidewall boundary.

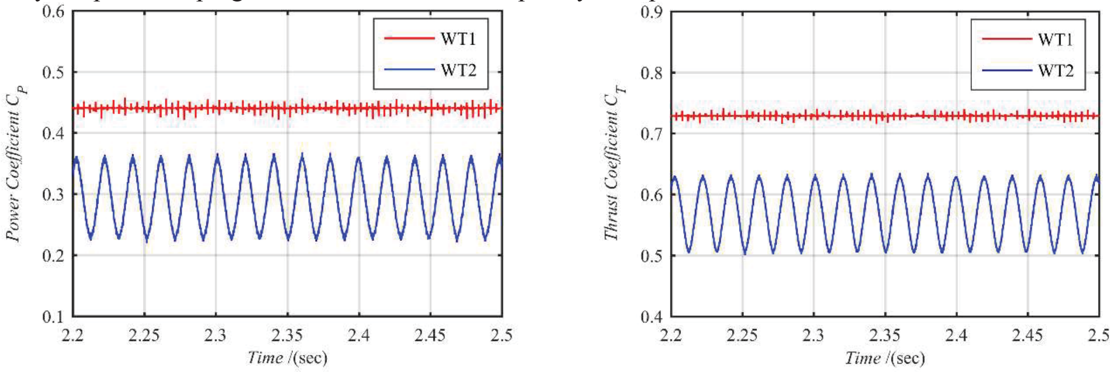
Results of Validation

The upstream wind turbine operates at tip speed ratio 6 and the downstream wind turbine runs at lower tip speed ratio 4.75. The experimental value from the "Blind Test 3" and the numerical value are summarized in table 2. From the validation of aerodynamic loads against the results from "Blind Test 3", it can be concluded that there is quite well prediction of aerodynamic load including the aerodynamic power and thrust for the upstream wind turbine when simulation for two offset wind turbines by using CFD technique the actuator line model. There is reasonable error for the engineering application, whose error is not over 5% for the both wind turbines.

Table 2. Summary of numerical and experimental results for aerodynamic power and thrust coefficient

Items		Numerical value	Experiment	Relative error
C_P	WT1	0.440	0.430	+2.326%
	WT2	0.292	0.299	-2.341%
C_T	WT1	0.801	0.771	+3.891%
	WT2	0.570	0.547	+4.205%

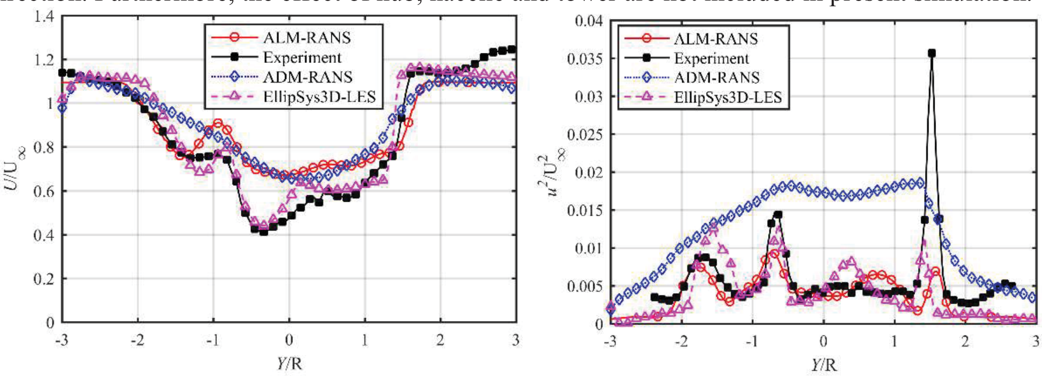
The time history curves of aerodynamic loads are shown in Figure 3. The most noticeable feature of these time history curves of aerodynamic loads, aside from the difference in loads levels, is that there is obvious oscillation in the loads curves of the downstream wind turbines. The loads curves of the wind turbine in the middle with the offset layout have a periodic variation that the frequency of oscillation equals to three times the rotor rotation frequency on these three-bladed turbines roughly. The reason of this interesting phenomenon may be that the middle turbine in the offset model is affected by the partial impingement of the wake developed by the upstream turbine.



(a) aerodynamic power (b) aerodynamic thrust

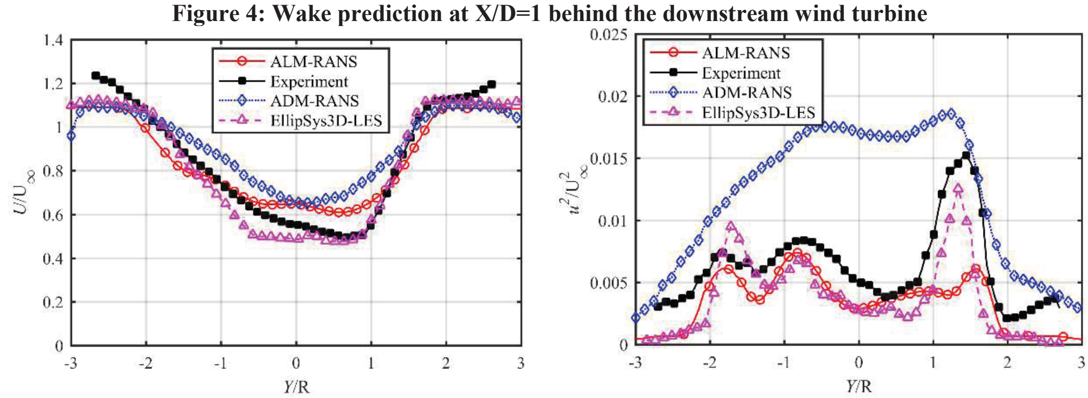
Figure 3: Time-history curve of aerodynamic loads.

Meanwhile, Figures. 4~5 show the wake prediction including the mean wake velocity and mean turbulent stress at two positions (X=1D, 3D; D is the rotor diameter of the downstream turbine) behind the downstream wind turbine along the width direction. Furthermore, the effect of hub, nacelle and tower are not included in present simulation.



(a) Mean wake velocity (b) Mean turbulent stress

Figure 4: Wake prediction at X/D=1 behind the downstream wind turbine



(a) Mean wake velocity (b) Mean turbulent stress

Figure 5: Wake prediction at X/D=3 behind the downstream wind turbine

From the figures, the numerical value by using the ALM is marked with solid red circles, there are a few obvious observations that may be made immediately. The mean wake velocity profile calculated from the simulation using actuator line model is higher compared to the experimental results marked in filled black squares. Since the current simulations don't take into account the effect of the nacelle and tower, the wake generated by the tower is ignored, which leading to the prediction for the wind velocity deficit is some lower. Furthermore, to a certain extent, the mean turbulent stress profile also has a good agreement with the experiment. Though the present study underestimates the turbulent stress magnitude especially in the peaks, whose value is much lower than the "Blind Test 3".

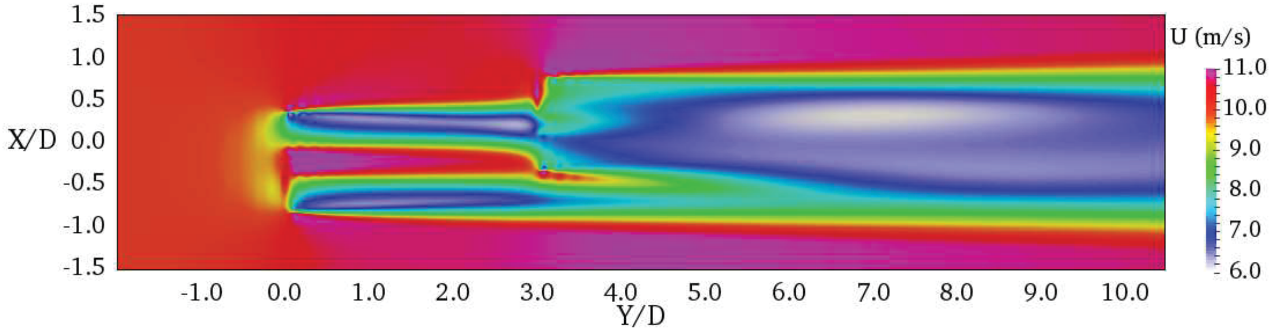


Figure 6: Axial velocity contour

Figure 6 shows contours of instantaneous streamwise velocity in horizontal plane through the center of wind turbine rotor, these figures are meaningful to understand overall wake flow regions. There is a higher speed region of flow near the center of the rotor. The reason for this phenomenon is that the effect of the hub, nacelle and tower are not considered in present simulation. It can also imply that the influence of upstream wind turbine on wake is significant, so there is still strong wake interference which is the main reason that causes the serious decrease in velocity of wake flow.

Conclusion

From above discussion, in actuator line simulations, the boundary layers are not explicitly simulated, but their effect is taken into account via the lift and drag coefficients. Although there is a certain error in wake prediction between the numerical simulation and the experiment "Blind Test 3", the overall change trend is consistent with the experiment. The actuator line model still can result the aerodynamic loads of the two offset model wind turbines and basic features of the wake.

Acknowledgements

The authors thank all those involved in the organisation of OFW13 and to all the contributors that will enrich this event. This work is supported by the National Natural Science Foundation of China (51490675, 51379125, 11432009, 51579145), Chang Jiang Scholars Program (T2014099), Shanghai Excellent Academic Leaders Program (17XD1402300), Shanghai Key Laboratory of Marine Engineering (K2015-11), Program for Professor of Special Appointment (Eastern Scholar) at Shanghai Institutions of Higher Learning (2013022), and Innovative Special Project of Numerical Tank of Ministry of Industry and Information Technology of China (2016- 23/09), to which the authors are most grateful.

References

[1] Abderrazzaq M A, Hahn B (2006). Analysis of the turbine standstill for a grid connected wind farm (case study) [J]. Renewable energy, 2006, 31(1): 89-104.

[2] Frandsen S T, Jørgensen H E, BARTHELMIE R, et al. The making of a second-generation wind farm efficiency model complex[J]. Wind Energy, 2009, 12(5): 445-458.

[3] Sørensen J N, Shen W Z. Computation of wind turbine wakes using combined Navier-Stokes/actuator-line Methodology[C]//1999 European Wind Energy Conference and Exhibition. 1999: 156-159.

[4] Churchfield M J, Lee S, Moriarty P, et al. A large-eddy simulation of wind-plant aerodynamics[C]// 50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition. 2012: 537.

[5] Churchfield M J, Lee S, Michalakes J, et al. A numerical study of the effects of atmospheric and wake turbulence on wind turbine dynamics[J]. Journal of turbulence, 2012 (13): N14.

[6] Ai Y, Wan D, Hu C. Effects of Inter-Turbines Spacing on Aerodynamics for Wind Farms Based on Actuator Line Model[C]//The 27th International Ocean and Polar Engineering Conference. International Society of Offshore and Polar Engineers, 2017.

[7] Sørensen J N, Shen W Z. Numerical modelling of wind turbine wakes[J]. Journal of fluids engineering, 2002, 124(2): 393-399.

[8] Pierella F, Eriksen P E, Sætran L, et al. Invitation to the 2012" Blind test 2" Workshop Calculations for two wind turbines in line[J]. Dept. Energy and Process Eng., NTNU, Trondheim, Norway, 2012.

[9] Krogstad P Å, Sætran L, Adaramola M S. "Blind Test 3" calculations of the performance and wake development behind two in-line and offset model wind turbines[J]. Journal of Fluids and Structures, 2015, 52: 65-80.