Research on Dynamic Stall and Aerodynamic Characteristics of Wind Turbine 3D Rotational Blade

HU Guo-yu, SUN Wen-lei, Dong Ping

The School of Mechanical Engineering
Xinjiang University
Urumqi, Xinjiang, China

Abstract — The aerodynamic behavior of wind turbine is of importance for the performance of wind turbine and thus is currently research focus. Based on computational fluid dynamics (CFD) method, this paper simulated the aerodynamic characteristics of NREL Phase VI wind turbine. The airflow distribution along the span of wind turbine blade was obtained by solving Unsteady Reynolds-Averaged Navier-Stokes (URANS). Extensive comparisons with NREL experimental data at constant pitch and variable wind speed show that the CFD predictions match the experimental data consistently well at low wind speed. At high wind speed, there are a few discrepancies due to the fact that the effects of flow separation lead to dynamic stall. The simulation results reveal the unsteady aerodynamic characteristics of wind turbine blade with three-dimensional rotational effect and validate the capabilities of CFD code.

Keywords - Wind Turbine; Three-Dimensional Rotational Effect; Unsteady Aerodynamics; Dynamic Stall

I. INTRODUCTION

Currently, wind energy as cost-effective and clean energy has been constantly strengthened. The development of wind turbine technology results in larger size and more complex wind turbine systems. The operational conditions of large-scale wind turbines are very complicated and therefore the aerodynamic behaviors are also more complex. The flow field around the blades has prominent characteristics of 3D rotational and unsteady effects. The coupling effect of dynamic stall and turbulences on aerodynamic behavior of wind turbine is remarkable. Dynamic stall behavior of wind turbine will increase aerodynamic loads on blades and therefore the effect of dynamic stall needs to be considered while calculating unsteady aerodynamic behavior in order to avoid underestimating aerodynamic loads. It is challenge for aerodynamic calculations of wind turbine containing complex dynamic inflow, turbulence, separated flow, dynamic wake and wake disturbance and so on. Compared to other analysis methods of wind turbine aerodynamics, CFD simulation method has significant advantages in dealing with 3D flow field and revealing 3D rotational effect. Therefore, CFD simulation method for aerodynamic behaviors of wind turbine will be the focus research field in the future.

The aerodynamics of wind turbines concerns modeling and prediction of the aerodynamic forces on the rotor blades of the turbine. There are several aerodynamics modeling codes for predicting aerodynamic performance of wind turbine at different stages of sophistication including Blade Element Momentum code (BEM), Wake codes and Full-3D CFD including turbulence modeling. Blade Element Momentum code has relatively high computational efficiency compared to wake codes and CFD. However, BEM codes is mostly used for 2-D airfoil data and steady conditions and hence they are not always sufficiently reliable for predicting the aerodynamic load with unsteady effects, particularly for stalled and yawed rotor conditions. Stall typically occurs at large angles of attack depending on the aerofoil. The boundary layer separates at the tip rather than further down the aerofoil causing a wake to flow over the upper surface drastically reducing lift and increasing drag forces [1].

Modeling of stall behavior is of utmost importance as was emphasized by Tangler and Kocurek [2]. Wind turbines are subjected to atmospheric turbulence, wind shear from the atmospheric boundary layer, wind directions that change both in time and in space, and effects from the wake of neighboring wind turbines. Full-3D CFD including turbulence modeling is a good choice for predicting aerodynamic behaviors with rotational and unsteady effects [3]. For the first time [4], CFD gave more accurate results and details of the flow and loads than ordinary BEM and vortex methods. However, predicting the stall behavior caused by rotational and unsteady effects is very complicated and required a sophisticated CFD model. Besides, large-scale turbines place more challenges for CFD to simulate the full scale 3D turbine with long blades and high Reynolds number.

Therefore, this paper used the incompressible CFD dynamic overset code to compute the aerodynamics of NREL Phase VI wind turbine rotor. Using k-ω turbulent model and Mann wind turbulence model recommended by the IEC 61400-1 ed. 3 [5, 6] as boundary and initial conditions, this paper developed CFD simulation methodology with complete simulation model of the rotor of wind turbine for predicting wind turbine aerodynamics. Extensive comparisons with UAE phase VI experimental data demonstrate the code has the ability to simulate wind turbine aerodynamics. The complete rotor geometry of wind turbine was modeled, including approximate geometries for
hub and blades. Unsteady Reynolds-Averaged Navier-Stokes equations were solved and k-ω turbulence models were used in the simulations. The simulation results for variable wind speed at constant blade pitch angle show that the CFD predictions match the experimental data consistently well, encompassing the general trends for thrust and power, sectional pressure coefficients and normal force coefficients at different sections along the blade.

II. COMPUTATION METHOD AND MODEL

A. Governing Equations

The incompressible and unsteady Reynolds-Averaged Navier-Stokes CFD code is used as the flow solver for the turbine simulations in this paper. According to Reynolds-Averaged Navier-Stokes method, the time mean as any primary control variables is defined as:

$$\overline{\Phi} = \frac{1}{\Delta t} \int_0^{\Delta t} \Phi(t)dt$$  \hspace{1cm} (1)

where the superscript “—” is the mean of time, \(\Phi^t\) is the fluctuating value of the variable. The instantaneous value of primary control variables can be written as the sum of mean and fluctuating values:

$$\Phi = \overline{\Phi} + \Phi^t$$  \hspace{1cm} (2)

The diameter of rotor studied in this paper is 10 m. The flow density is not changed and hence the compressibility of the flow is not considered due to low Mach number. The continuity equation can be written as:

$$\frac{\partial}{\partial x_i} (\rho u_i) = 0$$  \hspace{1cm} (3)

The control body momentum changes satisfy conservation of momentum theorem, expressed as:

$$\frac{\partial}{\partial x_i} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \frac{\partial}{\partial x_j} (-\rho u_i u_j)$$  \hspace{1cm} (4)

where \(u_i, u_j(i,j=x,y,z)\) is velocity component of x,y,z coordinate directions(m/s), \(-\rho u_i u_j\) is Reynolds stress, according to Boussinesq hypothesis, expressed as:

$$-\rho u_i u_j = -\rho \delta_{ij} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \delta_{ij} \frac{\partial \rho u_i}{\partial x_j}$$  \hspace{1cm} (5)

where \(\mu\) is turbulent viscosity coefficient, \(P_i\) is pressure caused by fluctuating velocity, defined as:

$$P_i = \left( \rho u_i^2 + v^2 + w^2 \right) = \frac{2}{3} \rho k$$  \hspace{1cm} (6)

\(\delta_{ij}\) is Dirac function, the mathematical expression is:

$$\delta_{ij} = \begin{cases} 1 & (i = j) \\ 0 & (i \neq j) \end{cases}$$  \hspace{1cm} (7)

In order to assure the above equations is closed, it is needed to add turbulence model.

B. Turbulence Models for CFD

Turbulence modeling is at the core of CFD. If the flow is laminar, CFD is capable of generating as accurate numerical solutions as desired, but this is not at all the case for turbulent flow. Therefore, it has to be modeled in an efficient way suitable for numerical solving of nonlinear partial differential equations of the transport equation type. A method well-known from quantum field theory and later applied to quantum and classical nonrelativistic many-body systems was also applied to turbulence, see [7]. While there is still some discussion about the range of validity to turbulence, its outcome was a modified k-\(\omega\) model [8]. This paper adopted k-\(\omega\) model and this model includes turbulent kinetic energy \(k\) and turbulent dissipation rate \(\omega\). Its transport equation is as following:

$$\frac{\partial}{\partial x_i} (\rho k) + \frac{\partial}{\partial x_j} (\rho u_j k) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\partial k}{\partial x_j} + \frac{\partial k}{\partial x_i} \right) \right] + \frac{\partial}{\partial x_j} (\mu \nabla^2 k) - \frac{\partial}{\partial x_j} (\rho \omega)$$  \hspace{1cm} (8)

$$\frac{\partial}{\partial x_i} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho u_j \omega) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\partial \omega}{\partial x_j} + \frac{\partial \omega}{\partial x_i} \right) \right] - \frac{\partial}{\partial x_j} (\mu \nabla^2 \omega) + C_w \frac{\omega^2}{k}$$  \hspace{1cm} (9)

where

$$S = \sqrt{2S_x S_y}$$

$$S_{ij} = \left( \begin{array}{c} S_{xx} \\ S_{yy} \\ S_{zz} \end{array} \right)$$

Turbulent viscosity coefficient is calculated by the following formula:

$$\mu_t = \rho C_{f} k^2$$  \hspace{1cm} (11)

The closed constants of this model are:

$$C_{f} = 1.44, \quad C_{s} = 1.92, \quad C_{\mu} = 0.09$$

$$\sigma_{ij} = 1.0, \quad \sigma_{ij} = 1.3$$

C. Wind turbulent model

An appropriate atmospheric wind turbulence input need to be provided in this simulation for dynamic wind loading analysis of wind turbine. Mann proposed and developed an efficient wind turbulence model based on the construction of a spectral tensor for atmospheric surface layer turbulence. Mann model has the same second-order statistics as in the real atmosphere and the ability to simulate three-dimensional fields with all three components of the wind velocity fluctuations.

In order to maintain the physical nature of wind turbulence and simplify its derivation, Mann turbulence model has a few assumptions that the wind velocities follow Gaussian distribution and the statistics of the atmospheric wind turbulence of the model are assumed to be homogeneous in space and stationary in time. Therefore, Mann model is stochastically consistent well with the
measurements of wind turbulence. Based on that, the Taylor’s frozen turbulence hypothesis is adopted in Mann model to relate the spatial wind fluctuations with time, which interprets time series as “space series”. In order to estimate the shear effect on the turbulence, Mann’s model linearizes the N-S equation on the basis of the isotropic Von Karman spectrum by assuming a uniform shear with the flow represent the mean wind field in the prevailing wind direction $U(x) = zdU/ezdz$ (where $dU/dz$ is a constant, $z$ is the vertical position, $\gamma$ is the prevailing wind direction).

III. SIMULATION OF UNSTEADY AERODYNAMICS OF NREL PHASE VI

The NREL Phase VI wind turbine was extensively tested in the NASA Ames wind tunnel in 2000. The main objective was to collect detailed load data from defined operating conditions in a wind tunnel, different and complementary to those gained in free field. Most computations to date testing numerical methods for aerodynamic predictions are compared to the National Renewable Energy Laboratory (NREL) Unsteady Aerodynamics Experiment (UAE).

The NREL Phase VI rotor is a two-bladed 10.1 m diameter wind turbine rotor with a rated power of 19.8kW (Figure 1). The blade geometry is based on the S809 aerofoil (Figure 2). Figure 3 and Figure 4 give the chord and twist distributions of the blade and 3D blade model respectively. One of the blades was equipped with pressure taps at 0.30R, 0.47R, 0.63R, 0.80R and 0.95R to acquire detailed surface pressure data. At each of the full pressure tap distributions, pressures were integrated to obtain $C_n$, $C_t$ and $C_m$. The blade was also equipped with five-hole pressure probes at 0.34R, 0.51R, 0.67R, 0.84R and 0.91R to measure the local inflow angle (LFA). Various loads were measured using strain-gauge techniques. These included the blade root flap and edge moments and the low-speed shaft torque.

The sequences S is selected from the test matrix of the NREL experiments as one case to compare to the simulation results for evaluating the ability of the code to predict the aerodynamics under different wind velocities (5, 10, 15 and 25 m/s) at a fixed 3 degrees blade tip pitch angle. The simulation process is shown in Figure 5.
The no-slip wall boundary condition was used on blades. Such parameters as wind speed and the scale of turbulence were set on the upstream inlet boundary and pressure outlet boundary was used on the downstream outlet.

B. Results and Discussion

The sequence S of NREL experiments is for variable wind speed at constant pitch angle. It comprises data for wind speeds from 5 to 25 m/s at intervals of 1 m/s at 3 degrees of blade pitch. Fixed pitch wind turbine avoid excessive aerodynamic loads at high wind speed by the large-scale separation (stall) on the boundary layer. For sequence S, experimentally measured power values will fluctuate in a relatively large range and flow phenomena is relatively complex. The CFD study is focused on 5, 10, 15 and 25 m/s wind speed and 3° pitch angle. The simulation for the highest two wind speeds is challenging because they correspond to stall conditions in most of the blade.

1) Trust and Power

Figure 7 show comparisons of CFD predictions and NREL experiments for thrust and power. As shown in Figure 7, CFD predictions of thrust and power are all well within the standard deviation of the experimental data, but there is a slight over prediction at 25 m/s where stalled flow and separation are remarkable. It is noticed that the flattening of the power as a function of the wind speed is predicted properly as the turbine becomes stall-controlled at higher wind speeds. The general performance of the turbine is predicted very well in this simulation, even at the two highest wind speeds where stalled flows occur.
HU GUO-YU et al: RESEARCH ON DYNAMIC STALL AND AERODYNAMIC CHARACTERISTICS OF …

Figure 7 Comparisons of thrust and power between CFD and NREL experiments

(a) Thrust

(b) Power

2) **Sectional Normal Force Coefficient $C_n$**

The sectional normal force coefficient directly influences the pressure distribution on blade sections. The computations of sectional normal force coefficient $C_n$ can have a better check on capabilities of the code to properly capture the aerodynamic behavior of wind turbine. This paper performed CFD simulation by solving Reynolds-Averaged Navier-Stokes (RANS) equation. Figure 8 gives the comparison of CFD prediction and experimental results of at 5 different sections ($r/R=0.30, 0.47, 0.63, 0.80, 0.95$) of the blade. As shown in Figure 8, CFD predictions based on RANS match very well the experimental data for all wind velocities simulated. At 5 m/s, there is no flow separation but at the transition section near the root. At higher wind velocities, where vortex shedding occurs and flow separation becomes more significant, discrepancies appear gradually. At 15 m/s where the flow separation around the blade is weak, RANS results are very close to NREL experimental results for $r/R<0.5$, but more different for $r/R>0.5$ where separation and vortex shedding are remarkable and differences in flow pattern are prominent. At 25 m/s, RANS CFD is well consistent with experimental results for all sections.

3) **Sectional Pressure Coefficient $C_p$**

Figure 9 compares the pressure coefficient between CFD predictions and experimental data at 5 different sections for 5 m/s and 10 m/s wind speed. At 5 m/s wind speeds the CFD predictions match the experimental data remarkably well. For 10 m/s at $r/R=0.3$ CFD predicts higher pressure than experimental measurements on the suction side, indicating unsteady flow and separation near the root, while CFD results predict a leading edge peak. Duque et al. [9] argued that the reason for these discrepancies is possibly unsteadiness. It can be seen from the results there is a relatively small pressure gradient along the blade chord as flow field interference is caused by the transverse flow on the blade. At $r/R=0.95$ where flow is attached, CFD results have good agreement with experiment data, indicating the adopted turbulent model can give details of the flow at the outside of the blade.
Figure 9  Pressure coefficient for 5m/s and 10 m/s at 5 radial sections
4) Flow Analysis on Suction Side of Blade

Figure 10 shows instantaneous limiting streamlines on the suction side of the blade for 5m/s, 15m/s and 25m/s velocities at 5 radial sections. At 5 m/s fully attached flow everywhere in the active blade is predicted in this simulation. CFD computations also predict some flow separation on the transient sections between S809 blade profile and a cylindrical section. At this speed the blade provides consistent lift and the pressure stays low on the suction side. At 15 m/s the flow separation is significant at all sections and there are smaller scale and unsteady vortices shed in the outer sections of the blade. At this speed for section r/R=0.47 the flow separation from the leading edge occurs and then reattaches to the surface of the blade to form a closed separation bubble. For all other sections, open separation occurs while vortices are shedding away from the blade surface to the wake. For the outer sections, it can be observed significant pressure recover. For all sections at 25 m/s, although a similar trend can be observed, the separation is stronger with violent vortex shedding and there is very little pressure recovery on the suction side indicating massive stall.

IV. CONCLUSION

According to NREL experiment test cases at constant pitch angle for variable wind speed, the unsteady aerodynamics of NREL Phase VI wind turbine is simulated based on computational fluid dynamics (CFD) method. Compared to NREL experimental measurements, the CFD predictions match the experimental data consistently well and validate the capabilities of the code for wind turbine simulations. It can be concluded that: 1) At low speed, $k-\omega$ SST turbulent model can well describe the aerodynamic characteristics for attached flow. 2) At high speed, $k-\omega$ SST turbulent model is insufficient for predicting the pressure of suction surface due to the effects unsteady vortex, but reveals the fact that separated flow on blade sections results in dynamic stall.

ACKNOWLEDGEMENTS

This work is supported by National Natural Science Fund Project, China (No. 51565055), Excellent PhD innovation projects in Xinjiang University, China (No. XJUBSCX-2010005).

REFERENCES