Analysis of wind turbine blades aeroelastic performance under yaw conditions

Liping Dai a,*, Qiang Zhou a, Yuwen Zhang b, Shigang Yao a, Shun Kang a, Xiaodong Wang a

a School of Energy Power and Mechanical Engineering, North China Electric Power University, Beijing, 102206, China
b Department of Mechanical and Aerospace Engineering, University of Missouri, Columbia, MO, 65211, USA

ARTICLE INFO
Keywords:
Horizontal axis wind turbines (HAWTs)
Aeroelastic performance
Yaw condition
Computational fluid dynamics (CFD)
Computational structural dynamics (CSD)

ABSTRACT
The aeroelastic modeling of Tjæreborg wind turbine blades was performed based on the unsteady Reynolds Averaged Navier-Stokes equations (URANS) combined with Finite Element Method (FEM) in a loosely coupled manner. This method was verified by comparing numerical and experiment results at four axial inflow wind speeds. Furthermore, the aeroelastic performance of Tjæreborg wind turbine under yaw angle of 10°, 30° and 60° were computed and analyzed. The results showed that the average power and thrust of the wind turbine decreased with increasing yaw angle, along with the increasing oscillation amplitude under large yaw angle. The aerodynamic load showed periodic change within one revolution of rotor, resulting in the blade deflection and the structure safety and instability. Besides, due to operating in complex wind with varying directions and magnitudes for horizontal axial wind turbines (HAWTs), or operating on the plunging platform for floating offshore wind turbines (FOWTs), it often happens that wind turbines work under yaw conditions. The understanding of aeroelastic behaviors under yaw conditions become increasingly urgent for large-scale wind turbine design.

The aeroelastic characters of wind turbine under yaw condition are a very complicated problem that requires integration of aerodynamic and structural dynamics models. Due to limitation of wind tunnel size and the complexity in measurement, there are not much experimental data of aeroelastic performance of wind turbines. Many aerodynamic experiments have been conducted in the prototypes (Jonkman et al., 2009; Hand et al., 2001) or the scale-down models (Noura et al., 2012; Ozbay; Verelst et al., 2012) in axial free stream or yaw conditions; these results presented abundant results for verification of theoretical methods or numerical methods. The theoretical and numerical methods of wind turbine aerodynamic performance mainly include Blade Element Momentum (BEM) (Lee et al., 2012), Lift Line Theory (LLT) (Farrugia et al., 2016), Vortex Lattice Theory Method (VLT) (Gebhardt and Roccia, 2014; Pesmajoglou and Graham, 2000), as well as computational fluid dynamics (CFD), such as Unsteady Reynolds Averaged Navier-Stokes equations (URANS) (Cai et al., 2016; Spentzos et al., 2005) and Large Eddy Simulation (LES) (Mehta et al., 2014). The BEM has been widely used in commercial software (GH Bladed, ADAMS, FAST, etc.) and in-house codes nowadays. It can give relatively accurate results with low computational cost at steady state condition, and later Modified BEM (MBEM) was presented to predict aerodynamic performance at unsteady state conditions (Ke et al., 2015a), in which all the related unsteady flow effects are considered. However, the CFD is a powerful tool to simulate unsteady flow field and structural response. Therefore, the coupled analysis of aerodynamic and structural models is needed for accurate prediction of aeroelastic performance of wind turbines.

1. Introduction
To alleviate energy crisis and meet the challenge to the environment, wind energy has been experiencing ever-increasing development and plays a more and more important role in the electric industry. Compared with decades ago, the rated power for wind turbines has increased from 50 kW to about 5 MW, and the diameter of rotor has also increased from 10 to 15 m to more than 100 m. The longer rotor blade with more flexibility leads to considerable blade deflection in flapping, lead-lag and torsional directions, which affect aerodynamic load distributions; the aerodynamics performance will in turn affect the structure safety and instability. Besides, due to operating in complex wind with varying directions and magnitudes for horizontal axial wind turbines (HAWTs), or operating on the plunging platform for floating offshore wind turbines (FOWTs), it often happens that wind turbines works under yaw conditions. Hence, the understanding of aeroelastic behaviors under yaw conditions becomes increasingly urgent for large-scale wind turbine design.

The aeroelastic characters of wind turbine under yaw condition are a very complicated problem that requires integration of aerodynamic and structural dynamics models. Due to limitation of wind tunnel size and the complexity in measurement, there are not much experimental data of aeroelastic performance of wind turbines. Many aerodynamic experiments have been conducted in the prototypes (Jonkman et al., 2009; Hand et al., 2001) or the scale-down models (Noura et al., 2012; Ozbay; Verelst et al., 2012) in axial free stream or yaw conditions; these results presented abundant results for verification of theoretical methods or numerical methods. The theoretical and numerical methods of wind turbine aerodynamic performance mainly include Blade Element Momentum (BEM) (Lee et al., 2012), Lift Line Theory (LLT) (Farrugia et al., 2016), Vortex Lattice Theory Method (VLT) (Gebhardt and Roccia, 2014; Pesmajoglou and Graham, 2000), as well as computational fluid dynamics (CFD), such as Unsteady Reynolds Averaged Navier-Stokes equations (URANS) (Cai et al., 2016; Spentzos et al., 2005) and Large Eddy Simulation (LES) (Mehta et al., 2014). The BEM has been widely used in commercial software (GH Bladed, ADAMS, FAST, etc.) and in-house codes nowadays. It can give relatively accurate results with low computational cost at steady state condition, and later Modified BEM (MBEM) was presented to predict aerodynamic performance at unsteady state conditions (Ke et al., 2015a), in which all the related unsteady flow effects are considered. However, the CFD is a powerful tool to simulate unsteady flow field and structural response. Therefore, the coupled analysis of aerodynamic and structural models is needed for accurate prediction of aeroelastic performance of wind turbines.
phenomena, such as the dynamic inflow, the dynamic stall and so on can be taken into account via some empirical models (Oggaard et al., 2015; Li et al., 1999; Zhang and Huang, 2011). However, the BEM method based on slipstream assumption is still questionable in its prediction accuracy of the complex unsteady aerodynamic loads.

The LLT/VLM established based on the assumption of potential flow and thin airfoil theory can give results that are more precise in unsteady flow compared with BEM; especially when it is combined with the free wake model. However, the computational cost is relatively high due to iterations to obtain the wake position in free wake method. Shen et al. (2011) investigated the wind turbine aerodynamic load distribution in wind shear flow with LLT, and observed that the reduced fatigue damage with individual pith control (IPC). Qiu et al. (Qiu et al., 2014) carried out research on blade aerodynamic load distributions in yawing and pitching with improved lift line theory that introduces a wake model comprising vortex sheet model and tip vortex model, and then compared against LLT with four other wake models. Hankin and Graham (2014) studied the aerodynamic load for a 5 MW HAWT from National Renewable Energy Laboratory (NREL) operating in an upstream rotor wake with unsteady VLM method, and compared with experiments conducted with 1:250 scale. Then the nonlinear vortex correction method (NVCM) was developed to consider airfoil thickness and viscous effect (Gebhardt et al., 2010). The core of the NVCM is modification of the sectional bound vortex strength according to the difference between sectional lift from the VLM and that from the table look-up procedure. Jeon et al. (2014) simulated a floating wind turbine operated in a turbulent wake state. They found the turbulent wake state (TWS) arisen when the floating platform is pitching in the upwind direction and the convection of the tip vortex plays an important role in governing of the behavior of the rotor in a TWS. Nowadays, NVCM shows great potential in predicting the aerodynamic performance and wake structure of the wind turbine from the viewpoint of computational cost and accuracy.

Compared with the above two methods, CFD can obtain the detailed flow features both inside the boundary layer and near-wake structure, such as transition position and separation point location, though it will be executed at high computational cost. In this method, all the complex flow phenomena including the dynamic inflow, the stall delay among others

<table>
<thead>
<tr>
<th>Name</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blade length/(m)</td>
<td>29.1</td>
</tr>
<tr>
<td>Flange distance from rotor axis/(m)</td>
<td>1.46</td>
</tr>
<tr>
<td>Blade tip chord/(m)</td>
<td>0.9</td>
</tr>
<tr>
<td>Taper linear/(m/m)</td>
<td>0.1</td>
</tr>
<tr>
<td>Twist linear/(deg/m)</td>
<td>0.333</td>
</tr>
</tbody>
</table>

Table 1
Principal parameters of Tjæreborg Wind Turbine.

Fig. 1. Comparison of azimuthal variation of sectional air loads and pitching moment for rotor-alone and full wind turbine configurations (Yu et al., 2013).
can be obtained without any empirical models; consequently, it tends to be used more and more widely with the development of computational power. Nowadays with the development of accelerating convergence technique, the computational cost of CFD has decreased continually, and sometimes can be the same order as LLT combined with free wake model. Tran and Kim (2016) studied unsteady aerodynamic performance of a FOWT at surge motion with the URANS equations and overlap mesh technique; the effect of surge oscillation frequency and amplitude on aerodynamic loads were obtained. Yu et al. (2013) investigated the unsteady aerodynamic performance of a stall-regulated HAWT NREL phase VI under yaw condition based on the URANS equations combined with unstructured overset mesh. They found that the blade loading showed a periodic fluctuation with lower magnitudes at the advancing blade side.

In structural dynamics, modal approach (Lago et al., 2013), multi-body dynamics (MBD) (Mo et al., 2015) and computational structure dynamics (CSD) (Ke et al., 2015b) are effectively and commonly used tools. Inmodal approach, the equations of motion can be obtained according to Hamilton's principle, combined with beam element analysis with 2–4 degree of freedom (DOF) and modal decomposition. The results from modal approach are generally in good agreement with the experimental data in linear geometrical deflection blade. Compared with modal approach with low computational cost, CSD method can establish the real blade model comprising many layers of fiber reinforced composite materials with necessary components such as shear web and root fixtures. It can present a precise internal stress distribution at a fairly expensive computational cost. For sake of simplicity, the complex blade geometry can be expressed in shell or solid geometry.

MBD is a moderate method to balance computational time and accuracy, and the blade can be discretized into super-elements connected through springs and hinges to each other. Hence, the aeroelastic characters of wind turbine can be carried out through the combination of one of the aerodynamic solvers and one of the structural solvers. Mo et al. (2015) investigated the NREL 5 MW aeroelastic performance of an offshore wind turbine blade using BEM theory with B-L dynamic stall for aerodynamic performance and MBD method for structural dynamic. They found that the fluid-structure interaction had significant effect on blade aerodynamic loads, and dynamic stall can cause more violent fluctuation for blade aerodynamic loads compared with steady loads. Wang et al. (2014) carried on a simulation of 1.5 MW baseline wind turbine with BEM theory combined with a non-linear beam theory, and the results showed that flapwise deflection was reduced compared with the results from the linear aeroelastic code FAST. Jeong et al. (2013) investigated the effects of yaw error on blade behaviors and dynamic stability using BEM combined with non-linear modal approach method, and found that the yaw misalignments adversely influenced the dynamic aeroelastic stability. Yu and Kwon (2014) predicted the aeroelastic response of a rotor-alone configuration and a full wind turbine configuration for NREL 5 MW HAWT with a coupled CFD-CSD method. They showed that rotor-tower interaction had considerable effects on blade aerodynamic load distributions when blades pass by tower.

In this paper, the aerodynamic loads and aeroelastic responses under axial and yaw conditions were investigated with commercial software ANSYS Workbench, in which URANS equations with k-ω SST turbulent model and FEM were selected based on the structured mesh. The yaw effect on aerodynamic loads will be analyzed, and the fluid structure interaction effect will be obtained through comparison between the results solved by the coupling and CFD solvers.

2. Tjæreborg wind turbine

The Tjæreborg wind turbine was built and experimented in 1987 in a wind farm located in Tjæreborg - a village on the west coast of Denmark.
Due to abundant experimental data, the Tjæreborg wind turbine was widely studied both in code verification and wind turbine performance analysis. It is a pitch-controlled and 3-bladed HAWT, and rotating clockwise as view from upwind. It has rotor diameter of 61.1 m and hub height of 61 m. The rated shaft power is 2200 kW at constant rotational speed of 22.36 rpm. The cut in and cut out wind speeds are 5 m/s and 25 m/s, respectively. The tapering blade is made of NACA4412-4443 airfoils with a linear twist and a linear taper. The principal rotor parameter is summarized in Table 1, and the other details can be referenced to (PF et al., 1994).

3. Mesh generation and computational method

3.1. Computational zone, boundary conditions, and mesh generation

According to the results (shown in Fig. 1) calculated in reference (Yu et al., 2013), the aerodynamic loads are almost the same for rotor-alone and full wind turbine configurations besides the region when the blade passing by the tower. To reduce the computational cost and time, only the blades were considered in this paper.

The computational zone shown in Fig. 2(a) is a cylindrical zone with 20D length and 10D diameter, where D stands for rotor diameter; the rotor is located at the center of the cylinder. The flow field zone was divided into two parts: the rotational and stationary zones, and the sliding mesh technique was used to exchange information between them. The entire flow field was meshed with hexahedral cells. The grids in rotating zone and around blade generated with O4H topology are shown in Fig. 2(b)-(c). To verify the grid independence, two sets of grids with total grid number of about 0.7M and 2.8M for each blade were adopted to simulate the flow field, and the periodic boundary conditions were applied between the rotor blades. The coarse mesh and the refined mesh around blade are shown in Fig. 2(c) and (d), respectively.

The inlet of the computational zone was set to be velocity inlet boundary condition, given three velocity components on each coordinate according to the yaw angle $\gamma$. The sidewall and the outlet of cylinder zone were set to be pressure outlet boundary condition. The blade surfaces were assumed non-slip wall. The interfaces between blade and fluid was set to be fluid-structure interface, on which the data can be transferred every iterative time step. The mesh movement due to blade deformation was taken into account with mesh deformation technique.

In general, the wind turbine blade is composed of blade shell and shear web, and both of them are composed of many layers of fiber composite reinforced materials (shown in Figs. 3 and 4). The mechanical properties of wind turbine blade are usually given in two ways: 1) The detailed mechanical properties and geometric dimension of each layer, such as elastic modulus at each direction, shear modulus and passion ratio $E_x, E_y, G_{xy}, \gamma_{xy}$; 2) The edgewise, flapwise and torsional stiffnesses distribution along blade span. For Tjæreborg wind turbine blade, the mechanical properties were given in the second way, so the blade material was assumed to be isotropic. The thickness of blade can be adjusted to meet the stiffness distribution in flapwise and edgewise directions. In this paper, the elastic modulus, shear modulus and shell geometry with varying thickness shell geometry were selected according to the spanwise distribution of edgewise, flapwise and torsional stiffnesses given in (PF et al., 1994). In the calculation, the blade root was assumed to be fixed and the blade tip was set to be a free end. The tetrahedron mesh in blade with the total mesh number of about 45,000 is shown in Fig. 5. The computed mode response and the comparison between the present and experimental natural vibration frequency are shown in Fig. 6 and Table 2. It can be seen the first and the natural frequency compare well with experimental data. The discrepancies may come from the inaccurate matching of the stiffness between numerical setting and experiment data.

3.2. Computational method

The Unsteady Reynolds Averaged Navier-Stokes equations (URANS) and $k-\omega$SST turbulent model in rotational frame and stationary frame were selected in rotational zone and stationary zone, respectively; the sliding mesh technique is used for data transferring between interfaces. To take the deformation of structure into account, mesh deformation technique was selected. The continuum and momentum equations for...
where $U_j$ and $W_j$ represent the flow field velocity and control volume boundary velocity related to mesh deformation; $\rho$, $S_u$, and $P$ denote density, source item and pressure tensor respectively. $\mu_{eff}$ is effective viscosity, which can be obtained from the following equation:

$$\mu_{eff} = \mu_t + \mu$$

where $\mu_t$ is the turbulent viscosity and it can be obtained through turbulent model. In this paper, $k-\omega$SST model, which is a combination of $k-\varepsilon$ model and $k-\omega$ model, was adopted. This mode can give highly accurate prediction of the onset and the amount of flow separation under adverse pressure gradients.

The finite volume method (FVM) with second upwind difference scheme for advection terms, central difference scheme for diffusion terms, and implicit difference scheme for transient terms were adopted to discretize the above equations. Dual-time method and co-located grid
with coupled solver for pressure and velocity were utilized in solving the discretized equations. Under relaxation is used to obtain a stable solution.

The governing equation for the rotating structure is as following (PF et al., 1994):

$$[M] \{\ddot{x}\} + ([G] + [C]) \{\dot{x}\} + ([K] - [K_{sp}]) \{x\} = \{F(t)\}$$

(4)

where \(\{x\}, \{\dot{x}\}, \{\ddot{x}\}\) represent node displacement vector, node velocity vector, and node acceleration vector, respectively; \([M], [G], [C], [K]\), and \([K_{sp}]\) are mass, gyroscopic damping, damping, stiffness matrix and spin softening matrix, respectively. The load vector \([F(t)\) includes aerodynamic force, centrifugal force, gravity, the angular rotational velocity force and the added mass force. The added mass force is caused by the fact that the particle has to accelerate some of the surrounding fluid, leading to an additional drag of the following form:

$$F_{VM} = \frac{1}{2} \rho_f \left( \frac{dU_p}{dt} - \frac{dU_f}{dt} \right)$$

(5)

where \(\rho_f\) is fluid mass around the particle; \(U_p\) and \(U_f\) are fluid and particle velocity respectively.

The gyroscopic matrix \([G]\) and the stiffness matrix due to spin softening \([K_{sp}]\) can be expressed as:

$$[G] = 2 \int \left[ \frac{\rho}{\rho} \right] [N]^T \{\omega\} [N] d\nu$$

(6)

$$[K_{sp}] = \int \left[ \frac{\rho}{\rho} \right] [N]^T [\omega]^2 [N] d\nu$$

(7)

where \([N]\) and \([\omega]\) are shape function matrix and rotational matrix associated with the angular velocity vector \([\omega]\), \(\rho\) is element density.

For more information on those matrix, it can be referred to (ANSYS, Inc.). All the coefficient terms can be obtained according to FEA. In transient dynamic analysis, the Newmark time integration method was used to solve these equations at discrete time points. The Newmark time scheme is

$$(\rho \dot{\phi})_n = (\rho \dot{\phi})_{n-1} + \Delta t \left( \delta \left( \frac{\partial (\rho \dot{\phi})}{\partial t} \right)_n + (1 - \delta) \left( \frac{\partial (\rho \dot{\phi})}{\partial t} \right)_{n-1} \right)$$

(8)

Or

$$\left( \frac{\partial (\rho \dot{\phi})}{\partial t} \right)_n = \frac{1}{\Delta t^2} (\rho \dot{\phi})_n - \frac{1}{\Delta t} (\rho \dot{\phi})_{n-1} + \left( 1 - \frac{1}{\delta} \right) \left( \frac{\partial (\rho \dot{\phi})}{\partial t} \right)_{n-1}$$

(9)

where \(\delta\) is time integration coefficient for Newmark method, and Subscript n and n-1 indicate the time step.

After the stable results had been obtained based on CFD solver, the coupling of fluid field solver and structure solver were done in a loosely coupling manner. This method is suitable for linear regime. For Tjarnbo wind turbine, the maximum of blade tip is about 0.5% blade length. Usually when the blade tip deflection is less than 10% blade the geometrically nonlinear effect can be neglected (Lv et al., 2015). The physical time step was set to be 0.035375s, corresponding to a rotor revolution angle of 4.5°. The data transfer between fluid zone and structure zone interfaces were done in two ways. For displacement transfer the Profile Preserving data transfer algorithm is selected, and the bucket surface mapping algorithm is used to generate mapping weight. Each target node is checked to see if it is in the domain of any of the source elements. For each source element in the bucket, the vector \((\bar{x})\) is found as

$$\{x^e\} = [N^e(\xi)] \{x^e\}$$

(10)

where \([N^e(\xi)]\) is the matrix of linear shape functions associated with the source element and \(\{x^e\}\) is the vector of global coordinates of element-local node. Weight-based interpolation and subsequent under-relaxation are used to evaluate the final data applied on the target side of the interface.

The general grid interface algorithm is selected for transferring conserved quantities such as force. In this algorithm, the weight contributions are evaluated for each control surface based on the associated source and target element surface areas. If the source side of the interface is completely mapped to the target side of the interface, then the resulting target values are globally conservative.

When the convergence errors were less than 10-4 and the monitored point displacement was stable or periodic, the computation was considered to be converged. Here the RMS relative to the previous-step solution is used to be the convergence error. The RMS is defined as:

$$\text{RMS} = \sqrt{\frac{\Delta_i}{\Delta_i}}$$

(11)

where \(\Delta_i\) is the normalized change in the data transfer value between successive iterations within a given coupling step, and is measured as:

$$\Delta_i = \frac{\Delta}{\left( \max(|\phi|) - \min(|\phi|) + |\phi| \right)}$$

(12)

The residual mean square for mass and momentum equations at yaw angle of 30°, 60° are shown in Fig. 7, and the displacement of blade tip is expressed in Fig. 8.

3.3. Validation of grid independence

Due to the complexity of flow field such as stall delay and divergence

![Fig. 7. Convergence curve of fluid flow equations.](image)

Fig. 7. Convergence curve of fluid flow equations.

![Fig. 8. Total displacement of blade tip.](image)

Fig. 8. Total displacement of blade tip.
phenomena, the solution of flow field is more sensitive to the grid than the structure. Therefore, the grid independence tests were performed for steady flow at four axial wind speeds with refined and coarse grids mentioned above. The comparison between the computed power and experimental data is shown in Fig. 9(a). It can be seen that the computed data is in good agreement with the experimental data. The maximum difference between the results obtained using refined grid and coarse grid is 19.22% and it occurred at wind speed of 5 m/s. This result from the ratio of lift to drag of airfoil is highly sensitivity to grid at low angle of attack. The second maximum percentage difference (less than 3%) occurred at the free stream wind speed of 15 m/s, which is due to the complex flow phenomena such as separation occurred at blade root are also highly sensitive to grid distribution. In case of intermediate wind speeds, the difference between two sets of grid can be neglected. Hence, taking into account the high computational cost for fine grid, the aeroelastic modeling was carried out based on the coarse grid.

The comparison of computed power with the CFD solver and the coupling solver are shown in Fig. 9(b). As the wind speed increases, the effect of FSI on power is more important. The differences in case of 5 m/s, 10 m/s, 14 m/s, and 15 m/s are -0.22%, 1.19%, 2.11%, and 4.16%, respectively. For modern large wind turbines, the aerodynamic center of blade element is designed to be close to the twist axis of blade so that the torsional deformation can usually be controlled to be less than 3°. Although there is only slight difference between the powers predicted by CFD and by the coupling solver, the latter still showed better prediction accuracy than the former. In conclusion, the URANS with k-ω SST turbulent model coupled with FVM is capable of predicting the aeroelastic performance of wind turbine.

4. Results and discussions

The aeroelastic performance of Tjæreborg wind turbine under yaw angles of 10°, 30° and 60° at speed of 10 m/s were simulated based on CFD coupled with CSD method. The definition of azimuth angle and blade number is shown in Fig. 10.

Fig. 10. Definition of azimuth angle and blade number.

![Fig. 10](image)

![Fig. 11](image)

Fig. 11. The azimuthal variations of wind turbine power with two solvers for γ = 0°, 10°, 30° cases.

![Fig. 12](image)

Fig. 12. The azimuthal variations of rotor axial thrust distribution with two solvers for γ = 0°, 10°, 30° cases.
4.1. Analysis of overall performance parameters

Fig. 11 shows the azimuthal variation of wind turbine rotor power for four yaw conditions ($\gamma = 0^\circ, 10^\circ, 30^\circ, 60^\circ$) by CFD solver and the coupling solver. Compared with the results at axial free stream wind, the average power under yaw angle of $10^\circ$, $30^\circ$ and $60^\circ$ have decreased by 3%, 19% and 84%, respectively. In yaw conditions, the velocity component of

---

**Table 3:** Relative change of normal force and tangential force.

<table>
<thead>
<tr>
<th>Angle</th>
<th>$F_n$</th>
<th>$F_t$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$10^\circ$</td>
<td>$1%$</td>
<td>$10%$</td>
</tr>
<tr>
<td>$30^\circ$</td>
<td>$4%$</td>
<td>$21%$</td>
</tr>
</tbody>
</table>

---

Fig. 16. Average tangential force distribution along spanwise direction.

Fig. 17. Unsteady tangential force distribution under different azimuth angles.

Fig. 18. Tangential force distribution at four typical azimuth angles under yaw angle of $30^\circ$. 

---

Fig. 13. Power distribution for the 1st blade at yaw angle of $30^\circ$.

Fig. 14. Axial thrust distribution for the 1st blade.

Fig. 15. Average tangential force distribution along spanwise direction.
oncoming wind normal to the rotor plane is decreased by the cosine of the yaw angle; however, the power variation will agree with \( \cos^2 \gamma \sim \cos^3 \gamma \) approximation due to the combined effect of wind component normal to the rotor plane and wind component aligned tangent to rotor plane. For the Tjæreborg wind turbine, the coupling solver predicts a higher average power and smaller oscillation amplitude than those by CFD solver.

The azimuthal variations of wind turbine axial thrust for three yaw angle cases with two solvers are shown in Fig. 12. Compared with the result at axial free wind, the thrusts decrease by 0.86% and 11.4% under yaw angles of 10° and 30°, respectively. It means that in yaw conditions the thrust is associated with \( \cos \gamma \sim \cos^2 \gamma \). The difference between thrusts obtained from CFD solver and the coupling solver can be up to 4–5% for all the cases. Hence, in the strength design of large-scale wind turbine, the FSI effect must be taken into account.

The azimuthal variations of power and thrust for the single blade (the 1st blade) under yaw angle of 30° are shown in Figs. 13 and 14, respectively. Both power and axial thrust present a \( 2\pi \) periodic oscillation under yaw condition, which means that the tangential force and normal force show asymmetric characters with respect to azimuth angle. The maximum power and the minimum power occur at about 120° and 300° azimuth angles, respectively; the maximum thrust and the minimum thrust occur at about 60° and 240° azimuth angles, respectively. The coupling solver predicts a higher average power and thrust than those with the CFD solver. Compared with the CFD solver, the coupling solver presents larger oscillation amplitude and almost the same oscillation amplitude for power and thrust.

4.2. Effect of yaw angle on aerodynamic performance

Figs. 15 and 16 show the averaged tangential force distribution along the spanwise direction. The relative change of normal force and tangential force compared with those at axial free stream is summarized.

---

Fig. 19. The spanwise distribution of relative velocity \( V_{rel} \) and inflow angle \( \Phi \) at typical azimuth angles.

Fig. 20. Limiting streamline distribution on suction surface under yaw angle of 30° at azimuth angle of (a) 0° (b) 90° (c) 180° (d) 270°.
in Table 3. It can be seen the tangential force decreases faster with the increase of yaw angle than the normal force, which is in consistency with the variation trend of power and thrust with the yaw angle.

Figs. 17 and 18 show the unsteady force distribution at four typical azimuth angles at yaw angle of 30°. It can be observed the forces at azimuth angles of 0° and 90° are greater than those at the other two azimuth angles. Tangential force at azimuth angle of 180° is greater than that at azimuth angle of 0°, which is opposite to the normal force behavior.

The aerodynamic forces are directly related to the distribution of angle of attack or inflow angle, as well as relative velocity. The inflow angle \( \Phi \) and the relative velocity \( V_{rel} \) can be calculated from:

\[
\Phi = \arctg \left( \frac{w}{u \cos \theta + v \sin \theta + \Omega r} \right) \tag{13}
\]

\[
V_{rel} = \sqrt{w^2 + (u \cos \theta + v \sin \theta + \Omega r)^2} \tag{14}
\]

where \( u, v, \text{and } w \) stand for the wind local velocity components on \( x, y \), and \( z \) coordinates respectively; \( \Omega \) represents rotational speed; \( r \) and \( \theta \) are local radius to rotor axis and azimuth angle respectively.

Fig. 19 shows the magnitude of \( V_{rel} \) and the inflow angle distributions at different azimuth angles. At zero azimuth angle, the blade is advancing toward the velocity component aligned tangent to the rotor plane, and thus a higher magnitude of relative velocity and a lower angle of attack can be observed. On the other hand, at azimuth angle of 180°, the blade is retrieving the tangent component of inflow velocity, and a lower magnitude of relative velocity and a higher angle of attack can be found. Neglecting the change of pitch angle due to torsional deformation, the trend of angle of attack relative to azimuth should be identical as the inflow angle. Consequently, the angle of attack usually reaches the maximum and the minimum at azimuth angles of 180° and 0°, respectively. Hence, it can be inferred that the normal force is dictated more by the change of relative velocity, whereas the tangential force depends more on the change of angle of attack.

Because of combined effect of angle of attack and relative velocity, the normal force is greater at zero azimuth angle than that at azimuth angle of 180°. The differences between aerodynamic forces at azimuth angles of 90° and 270° are presumably related to the velocity component aligned tangent to rotor plane.

Fig. 20 shows the instantaneous limiting streamline distribution on suction surface for four typical azimuth angles. It can be observed that the flow is closely attached to the blade surface except blade root region. The tangential wind velocity component will produce inward flow at the azimuth angle of 90° and outward flow at the azimuth angle of 270°, and then the separation region will be shortened or prolonged. At azimuth angle of 0° and 180°, the separation region is small and large due to the angle of attack mentioned above. Therefore, at azimuth angle of 180° and 270°, separation region is characterized by separation line and reattachment line. In the other two azimuth angles, only separation lines can be found. Besides, because of the centrifugal force and Coriolis force pointing to outboard, the radial flow at azimuth angle of 270° is slightly distinguished than that at azimuth angle of 90°, which results in less aerodynamic forces at azimuth angle of 270° than those of 90°. From inboard to outboard, the difference of aerodynamic forces between azimuth angle of 90° and 270° are more and more noticeable, which means the radial flow has greater effect on the outboard than on the inboard.

The unsteady chordwise pressure coefficient distributions at four typical azimuth angles under yaw angle of 30° with the CFD solver are presented in Fig. 21. The pressure coefficient \( c_p \) is defined as following:

\[
c_p = \frac{(p - p_a)}{q} \tag{15}
\]

\[
q = \frac{1}{2} \rho \left( w^2 + (u \cos \theta + v \sin \theta + \Omega r)^2 \right) \tag{16}
\]
where \( p \) and \( p_{\infty} \) are the local pressure and pressure of incoming wind. It can be seen from Fig. 22 that in all blade span sections, the aerodynamic loads are characterized by the highest value at 180° azimuth and the lowest value at 0° azimuth, where the aerodynamic load is inferred from the difference of upper and lower pressure lines. It is consistent with the angle of attack distributions inferred from Fig. 19. The dimensionless aerodynamic load at 90° is slightly larger than that at azimuth angle of 270°, which may stem from the tangential wind velocity component mentioned above.

Fig. 22 shows the azimuthal variations of tangential force and normal force at 70% spanwise location, which is selected because it usually has great contribution to the power output. It can be seen that tangential force reaches the maximum at azimuth angle of 108° and the minimum at azimuth angle of 288°, and the normal force reaches the maximum at azimuth angle of about 72° and the minimum at azimuth angle of about 252°. The azimuthal variation of aerodynamic force is a combined effect of changes of angle of attack, relative velocity magnitude, and orientation of velocity component tangential to rotor plane. For different wind turbines operating in yaw condition, the azimuthal variations of tangential force and normal force can be different. For example, for NREL Phase VI in (Jeon et al., 2014), lower magnitude of both the tangential and normal forces could be found when the blade is advancing toward the wind component aligned tangent to the rotor disk plane and higher magnitudes in the remaining region. For NREL 5 MW wind turbine (Wang et al., 2014), a different azimuthal variation of aerodynamic loads were presented, where the higher magnitude of normal force and the lower magnitude of tangential force can be observed in region where the blade is advancing toward the wind velocity component aligned tangent to the rotor disk plane.

4.3. Effects of FSI on aerodynamic performances

Fig. 23 shows the comparison of the tangential and normal forces obtained from the CFD and the coupling solvers. The most distinguished difference takes place at blade outboard at azimuth angles of 90° and 180°, indicating the location of significant torsional deflection. At other azimuth angles, FSI has no significant effect on aerodynamic load distribution.

Fig. 24 shows comparison of chordwise pressure coefficients at 70% span section obtained by CFD and the coupling solvers under yaw angle of 30°. It can be seen that the differences of the pressure coefficient between two solvers at azimuth angles of 90° and 180° are more distinguishable than those at the other azimuth angles. At azimuth angles of 90° and 180°, the coupling solver predicts a slightly forward suction pressure peak compared with the results of the CFD solver, which indicates that a negative twist (nose up) and an increasing angle of attack occurred in the flow field.

Fig. 25 shows comparison of azimuth variation of tangential and normal force for 70% span section with two solvers. It can be seen that...
the maximum differences (15% for tangential force and 9% for normal force) between two solvers occur at azimuth angle of about 90°. In addition, the yaw angle will intensify the effect of FSI on aerodynamic loads due to the increasing maximum aerodynamic load.

Fig. 26 shows the power distribution for each blade separately. As discussed above, compared with the power obtained from the CFD solver, the coupling solver gives a higher power output for azimuth range between 45° and 216°, and less output in the other azimuth range for the 1st blade. For the 2nd and the 3rd blades, the same power distribution pattern with phase angle lag of 120° and 240° can be found.

4.4. Effects of yaw angle on blade deflection

The maximum deflection usually occurs on blade tip, and the flapwise and lead-lag deflections of blade tip are shown in Fig. 27. It can be seen that both the flapwise and lead-lag deflection show considerably asymmetric in yaw condition, with the maximum deflection occurring at azimuth angle of about 90° and the minimum deflection occurring at about 270°; this is similar to the aerodynamic force distribution pattern. The trend of flapwise deflection is similar to the lead-lag deflection; however, the flapwise deflection between 0.91 and 1.28 m is one order of magnitude higher than lead-lag deflection between 0.085 and 0.123 m. Besides, the maximum lead-lag and flapwise deflection increase with increasing yaw angle.

The distribution of the total deflection under yaw angle of 30° yaw angle condition is shown in Fig. 28. It can be observed that the total deflection also shows asymmetrical characters. The maximum deflection occurs at azimuth angle of 90° on the blade tip, and the minimum deflection occurs at azimuth angle of about 270° on the blade root. From tip to root, a nonlinear deformation is presented with largest deformation magnitude.
gradient occurred in mid-span zone and the smallest deformation gradient in blade root. This results in the stress distribution displayed in Fig. 29. The stress on y direction is one order magnitude higher than the other directions. The maximum stress appears in the leading edge of 50%-85% span at azimuth angle of about 90° on Y direction, which is greater than the maximum stress in the axial free inflow.

5. Conclusions

The aeroelastic modeling of wind turbine was performed using the URANS equations with k-ω SST turbulent equations for flow field and finite element method for structure; it was validated via the comparison between computational power and experimental power of the Tjæreborg wind turbine. The aeroelastic performances of Tjæreborg wind turbine under yawed condition were analyzed and the following conclusions can be drawn:

1) Compared with the axial free inflow, the average power of wind turbine in yaw conditions will decrease by \( \cos^2 \gamma \sim \cos^3 \gamma \); the averaged thrust agrees well with \( \cos \gamma \sim \cos^2 \gamma \). All parameters including inflow angle and relative velocity magnitude show asymmetric distributions along azimuth, which results in the maximum and the minimum aerodynamic loads occurring at azimuth angle of about 90° and 270°, respectively.

2) FSI has significant influence on aerodynamic load. For Tjæreborg wind turbine, the coupling solver gives higher average power and thrust, as well as more violent oscillation amplitude compared with those of the CFD solver. At wind speed of 10 m/s, the axial thrust increases up to 4% due to FSI.

3) Affected by the aerodynamic loads, the deflection of blade and the stress present asymmetrical distributions along the azimuth and non-
the maximum stress occurs at azimuth angle of about 90° and flange on blade leading edge.

Stress distribution on each direction under 30° apwise occurs at azimuth angle of about 90° yawed condition. FL/C14 on blade tip, and FL/C14 flange on lead-lag on mid-span FL/C14.

Acknowledgments

Support for this work by the Fundamental Research Funds for the Central Universities (2015MS37) and National Natural Science Foundation of China (51576065) is gratefully acknowledged.

References

ANSYS, Inc.: ANSYS Mechanical APDL Structural Analysis Guide.
Ahrmet Ozbay. An Experimental Investigation on Wind Turbine Aeromechanics and Wake Interferences Among Multiple Wind Turbines, Ph.D. Thesis, Iowa State University, USA.
Tran, Thanh Toan, Kim, Dong-Hyun, 2016. ACFD study into the influence of unsteady aerodynamic interference on wind turbine surge motion. Renew. Energy 90, 204–228.

Fig. 29. Stress distribution on each direction under 30° yawed condition.


