Challenges in Simulation of Aerodynamics, Hydrodynamics, and Mooring-Line Dynamics of Floating Offshore Wind Turbines

D. Matha and M. Schlipf
Universität Stuttgart

A. Cordle
Garrad Hassan and Partners Ltd.

R. Pereira
Germanischer Lloyd Industrial Services

J. Jonkman
National Renewable Energy Laboratory

Presented at the 21st Offshore and Polar Engineering Conference
Maui, Hawaii
June 19-24, 2011

NREL is a national laboratory of the U.S. Department of Energy, Office of Energy Efficiency & Renewable Energy, operated by the Alliance for Sustainable Energy, LLC.

Conference Paper
NREL/CP-5000-50544
October 2011

Contract No. DE-AC36-08GO28308

Available electronically at http://www.osti.gov/bridge

Available for a processing fee to U.S. Department of Energy
and its contractors, in paper, from:

U.S. Department of Energy
Office of Scientific and Technical Information

P.O. Box 62
Oak Ridge, TN 37831-0062
phone: 865.576.8401
fax: 865.576.5728
email: mailto:reports@adonis.osti.gov

Available for sale to the public, in paper, from:

U.S. Department of Commerce
National Technical Information Service
5285 Port Royal Road
Springfield, VA 22161
phone: 800.553.6847
fax: 703.605.6900
email: orders@ntis.fedworld.gov
online ordering: http://www.ntis.gov/help/ordermethods.aspx

Cover Photos: (left to right) PIX 16416, PIX 17423, PIX 16560, PIX 17613, PIX 17436, PIX 17721

Printed on paper containing at least 50% wastepaper, including 10% post consumer waste.

Challenges in Simulation of Aerodynamics, Hydrodynamics, and Mooring-Line Dynamics of Floating Offshore Wind Turbines

Denis Matha, Markus Schlipf
Endowed Chair of Wind Energy, Universität Stuttgart
Stuttgart, Germany

Andrew Cordle
Garrad Hassan and Partners Ltd.
Bristol, United Kingdom

Ricardo Pereira
Germanischer Lloyd Industrial Services, Renewables Certification
Hamburg, Germany

Jason Jonkman
National Renewable Energy Laboratory
Golden, Colorado, USA

ABSTRACT

This paper presents the current major modeling challenges for floating offshore wind turbine design tools. It also describes aerodynamic and hydrodynamic effects due to rotor and platform motions and usage of non-slender support structures. The applicability of advanced potential flow and computational fluid dynamics–based aerodynamic and hydrodynamic simulation methods to represent these effects—exceeding state-of-the-art design tool capabilities—is analyzed and the results are presented. Different techniques for the representation of mooring-line dynamics, including quasi-static, finite element, and multibody methods, and their impact on global system loads are investigated. Conclusions are drawn about the importance of the relevant effects, strengths and weaknesses of the different methods are discussed, and development needs of future tools are described.

KEY WORDS: Offshore; floating wind turbine; integrated design tools; mooring system; aerodynamics; hydrodynamics; potential flow; computational fluid dynamics; CFD; multibody simulation; MBS.

INTRODUCTION

Combined aerodynamic, hydrodynamic, and mooring-system dynamic effects on floating offshore wind turbines (FOWT) create unique operating and failure design conditions which have not yet been studied in great detail. The large rotor and platform motions and the use of non-slender support structures potentially render state-of-the-art techniques applied for modeling fixed-bottom offshore wind turbines—for example the blade element momentum (BEM) method for rotor aerodynamics and Morison's equation for hydrodynamics—insufficient for accurately describing the dynamics of floating wind turbines. Addressing these limitations and effectively designing and analyzing wind turbines on offshore floating support structures requires advanced modeling methods and techniques.

Dedicated hydrodynamic and mooring-system codes and techniques developed for the oil and gas industry for analysis of ships and oil platforms, as well as advanced aerodynamic methods used in the aircraft and helicopter industry, which are capable of addressing most effects relevant for FOWTs are not readily applicable for integrated FOWT simulations. The specific coupled aero-servo-hydro-elastic dynamics of FOWT and the requirements of the International Electrotechnical Commission (IEC) standard and certification guidelines (2009) to run large numbers of design load simulations must be considered when analyzing the application of existing and novel methodologies.

In sum, this paper describes the important physical effects unique for FOWTs in theory and the current major modeling challenges in FOWT design codes. It introduces techniques and methodologies to overcome the limitations of current integrated simulation tools, discusses their applicability for the simulation and analysis of FOWTs, and presents first indicative results of new approaches, including computational fluid dynamics (CFD) aerodynamic models and multibody system (MBS) mooring-line representations. This work also draws conclusions about the importance of the relevant effects, strengths, and weaknesses of the different methods and notes the development needed.

AERODYNAMICS

Aerodynamic Effects on Floating Wind Turbines

All of the design codes currently capable of performing integrated modeling of floating wind turbines (Cordle and Jonkman, 2011) use the Blade Element Momentum (BEM) theory to calculate the aerodynamic forces on the wind turbine blades. The significant low-frequency platform motions experienced by floating offshore wind turbines, however, result in flow conditions which are considerably different and more complex than those experienced by conventional onshore or fixed-bottom offshore wind turbines. Extensive load-case simulations performed by Jonkman, Matha, and Robertson (Jonkman, 2007; Matha, 2009; Jonkman and Matha, 2011; Robertson and Jonkman, 2011) have shown that, particularly for a catenary moored floating wind turbine, significant rotational pitch motions occur at approximately the incident-wave frequency.

For the investigated OC3-Hywind spar buoy featuring the NREL 5-MW baseline wind turbine, pitch motions of up to 8° for production design load cases (DLCs) and 14° for extreme DLCs have been found. In addition to significant pitch motions, large lower-frequency translational surge motions also are predicted. These rotational and translational motions often cause the rotor to operate in non-axial flow conditions, and lead to a change in the interaction between the rotor and wake with the rotor potentially traversing back over its own wake. The transitions between windmill and propeller states where the rotor interacts with its own wake cannot be accurately modeled using traditional BEM theory with common corrections, as Sebastian and Lackner (2010; 2011) have shown. A BEM theory cannot model the resulting development of a turbulent region behind the rotor, leading to toroidal recirculation normal to the rotor blade which is expected to be most significant at the blade tips in below-rated wind conditions. Sebastian and Lackner identify this transitional aerodynamic phenomenon as vortex ring state (VRS), and conclude that momentum equations used in typical BEM analysis methods yield unrealistic results for the axial and the rotational motion of the rotor.

Wind turbine airfoils in production up to rated power usually operate close to their maximum lift coefficient to generate maximum power, therefore the described pitch motions also cause the airfoil to operate in stalled condition more often, increasing the importance of dynamic stall models. Common dynamic stall models such as Beddoes-Leishman are semi-empirical and their applicability for the dynamic-stall effects occurring on floating wind turbines has yet to be investigated.

For floating support platform designs with relatively little yaw stiffness, yawed inflow conditions also occur more often. Yawed inflow also likely will occur frequently for proposed downwind floating concepts that don't include a tower-top mounted yaw drive (such as Sway's concept). The BEM theory as originally developed assumed flow perpendicular to the rotor plane, and commonly introduced skewed-wake correction models have yielded unreliable load predictions, with deviations increasing with greater yaw errors. The aforementioned proposed downwind FOWT configurations combined with the increased wind turbine motions also increase the importance of tower dam and shadow models, as well as improved models for aerodynamic blade and tower interaction.

The expected large ratings (10 MW and greater) of FOWT will lead to longer blades and greater tip speeds, due to (likely) reduced noise-emission regulations far offshore. Although beyond the scope of this paper, the aeroelastic representation of such large rotors also is important when investigating the aforementioned effects, and has significant influence (Matha, Hauptmann, Hecquet, and Kühn, 2010) on the general capability to predict some of these effects and the resulting loads, deflections, and instabilities (e.g., flutter).

Computational Fluid Dynamics

Several possibilities exist for modelling the described aerodynamic effects. The approach that computationally is most demanding, but physically is most accurate, is to solve the Navier-Stokes Equations (NSE) with a CFD solver. In CFD, to model turbulent flows with their large range of length scales, three basic approaches exist.

Direct numerical simulation (DNS), in which the full Navier-Stokes Equations are solved, requires all relevant length scales to be resolved by the computational grid. This method results in extremely large grids. Therefore, with current available computational power, DNS is not applicable to structures such as wind turbines with large Reynolds numbers.

Large eddy simulation (LES), in which only the equations for larger turbulent structures are resolved, is applicable for the turbine wake. In the boundary layer of the blades, LES also currently is very difficult to apply due to the great number of necessary grid cells.

The least computationally expensive method to model turbulent flows in CFD is the Unsteady Reynolds Averaged Navier-Stokes (URANS) method. The URANS method uses a time-average formulation of the Navier-Stokes Equations. The occurring nonlinear Reynolds stress term requires the introduction of turbulence models (for example, two-equation models such as k-ω) to close the URANS equations. These turbulence models can have great effects on the CFD solution and must be selected carefully. A combination of URANS and LES, using URANS for the flow around the blades and LES for the wake, also is possible and could provide a good solution for floating wind turbine CFD modeling.

Regardless of the specific CFD method used, due to very long resulting simulation times it can be applied only to very specific load situations. The primary application of CFD is to investigate and analyze the aerodynamic flow phenomena occurring in the situations noted above and help quantify their influence on loads. This knowledge can help in deciding whether current models still are sufficient and, more importantly, the significance of their errors and in what load situations these errors become most significant. Based on that knowledge, simpler aerodynamic codes suitable for integrated design codes—such as correction models for BEM—could be introduced, correction factors be derived, or new additions to the IEC standard (including novel load case definitions unique for floating turbines) be defined.

The studies by Jonkman (2007) and Matha (2009) also show that, for extreme DLCs with occurrence of failures (such as in DLC6.2) where the rotor is idling, all blades are pitched to feather, and great yaw errors occur, severe instabilities can be identified. The particular DLC6.2 instability is caused by negatively damped modes due to the blade aerodynamics, which are, for idling or stand-still conditions, being calculated with no induction factor (airfoil data look-up table). In this case, the aerodynamic lift and drag forces on the blade segments are computed without taking into account the blade's influence on the flow. To investigate this effect—as well as other observed instabilities for floating turbines—more closely, CFD also can provide valuable insight.

CFD methods already are used in wind turbine blade design and also can be applied to develop floating offshore–specific blades. Although, to date, no dedicated research on FOWT-specific blade design using a CFD process chain has been published, the European KIC (Knowledge & Innovation Community) work package, "Offshore Blades," of the InnoEnergy research project Offwindtech (started at the end of 2010) will address this issue.

Potential Flow Methods

More suitable for integrated load simulation in terms of simulation time are aerodynamic codes based on vortex theory. In potential flow theory, the fluid is considered to be incompressible, inviscid, and irrotational, and surface-tension effects are neglected. The flow field around the airfoil generally is described through the distribution of discrete sources and vortices, with several possible implementations, for example lifting line, lifting surface, or vortex lattice methods. In these time-accurate aerodynamic codes, the shape and strength of the wake of the blades will develop in time (e.g., free wake particle method). To reduce simulation time, the shape of the wake also can be prescribed. This approach, like BEM, is based on measured profile data. That is, the aerodynamic lift- , drag- , and pitching-moment characteristics of the blade cross-sections are assumed to be known and corrected for the effects of blade rotation. In comparison to the currently used BEM-based codes, more accurate predictions are expected in situations where local aerodynamic characteristics strongly vary with time—such as in yawed flow—and where dynamic wake effects play a significant role; effects which are increasingly important for floating wind turbines.

Currently there are several potential flow aerodynamic codes for wind turbine application in development. Specific analyses with these codes addressing the above-mentioned aerodynamic problems regarding floating wind turbines currently are performed by Sebastian and Lackner (2011) using an in-house free-wake code, and by Matha and others at University Stuttgart.

Rotor-Only Computational Fluid Dynamics Study

To investigate the aerodynamic effects occurring on a rotor of a FOWT, a full three-dimensional (3D) CFD model of a generic multi-megawatt rotor has been set up in the URANS code FLOWer (a URANS/LES CFD solver for structured meshes, developed by the German Aeronautical and Aerospace Centre DLR). Figure 1 presents the mesh used and the prescribed motion of the rotor. Note that the spinning direction of this particular rotor is counterclockwise when looking downwind. The prescribed motion was selected from the IEC load simulations of the OC3-Hywind FOWT conducted by Matha and Jonkman (Matha, 2009) in the design code FAST with AeroDyn and HydroDyn. A representative extreme platform-pitch motion occurring in wind turbine production mode in DLC 1.6 was chosen. The selected specific DLC 1.6 simulation run featured 12.0 m/s hub-height wind speed (close to rated, or maximum rotor thrust) and, using the extreme sea state (ESS) model, a significant wave height of 15.0 m and a peak spectral period of 19.2 s. One pitch motion was selected, where the rotor first is pitching 11.5° in downwind direction and then 4° in upwind direction before reaching 0° again. This motion has been approximated by two appended sine functions, as presented in Fig. 1 and also incorporates the associated surge motion (rotation is defined around an horizontal axis 90m vertically below rotor hub).

Following the work of Streiner (Streiner, Hauptmann, Kühn and Krämer, 2008), a sufficient timestep size corresponding to an azimuth movement of $\Delta\Psi = 5°$ per timestep was chosen. Before starting the CFD calculation of the prescribed floating motion, four rotor rotations were pre-calculated to ensure that transients had decayed and the wake behind the rotor was developed in the background grid when the motion starts. Due to good experiences in former studies by Streiner, the k-ω SST turbulence model was used. A uniform inflow velocity of 10 m/s was applied.

The computational grid consists of the components blade, hub, and background grid, equaling a total number of ~10 million grid points. Refined grids around each blade (Fig. 1, green grid), with a dimensionless grid distance y^+-value between 0.1 and 2 are embedded in a cylindrical grid around the hub (Fig. 1, red grid), which is rotating within a coarser background grid (Fig. 1, blue grid). The grid has the total in-plane dimension of 3 rotor diameters in each direction, centered at the hub, and a total out-of-plane dimension of 3.5 diameters upwind and downwind, respectively. FLOWer features the Chimera technique (DLR Braunschweig, 2007), allowing for arbitrary relative motion of aerodynamic bodies. Applying the Chimera technique, the previously described pitch motion was prescribed as a rigid-body motion on the rotor and hub grids, which are moving within the background meshes. Chimera also is applied for modeling the rotor rotation.

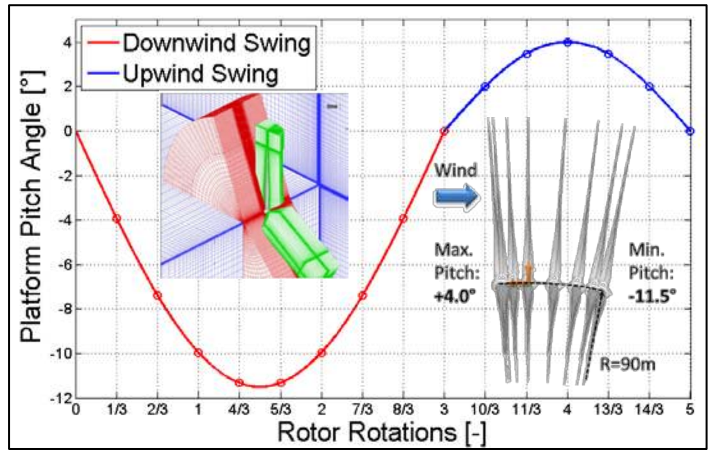

Fig. 1. Prescribed rotor motion for CFD study and CFD mesh

In this study, no elastic deformation of the blade is introduced, but fluid-structure coupled simulations are planned for subsequent analyses. Further studies also will include the tower and nacelle, as performed by Meister and others using FLOWer for a 5-MW onshore turbine under unsteady inflow conditions, but without prescribed floating motions (Meister, Lutz and Krämer, 2010).

The presented CFD results showed good numerical convergence, therefore it is assumed that the presented flow field is realistically representing the actual aerodynamic conditions during such a motion. Nevertheless, the results presented are from a preliminary study on a relatively coarse mesh and require further validation from CFD and potential flow calculations and, ultimately, experimental data.

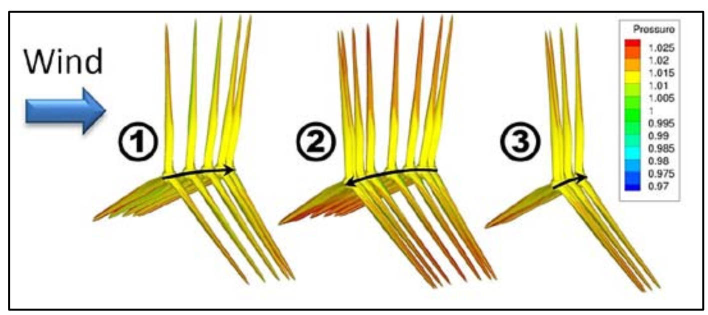

Fig. 2. Pressure contours of rotor during prescribed motion

The dimensionless pressure distribution over the blades is presented in Fig. 2. Motion (1) corresponds to the first half of the downwind swing (*c.f.* Fig. 1), which is 4/3 rotor rotations. Motion (2) corresponds to the second half of the downwind swing and the first half of the upwind swing, or from 5/3 to 4 rotor rotations. Motion (3) represents the last half of the upwind swing, 13/3 to 5 rotor rotations. During motion (1), the pressure on the pressure side of the blades decreases due to the backward motion and the increased turbulence in the wake. When the turbine is pitching upwind again, motion (2), the pressure on the blades increases until the rotor returns to its vertical position and then slightly decreases until the end of motion (2). The pressure increase is due to the increased inflow velocity, and also possibly due to rotor wake interactions. In the last part of the period, motion (3), the pressure again is decreasing, but to a lesser extent than in the first downwind motion (1).

Figure 3 presents the axial velocities in the near wake of the rotor in a vertical plane through the hub center. The single plots display the flow field after consecutive full rotor revolutions during the pitching motion, therefore covering the full movement from Fig. 1. Blade one in these plots always points upward and the axial flow velocity is directed to the right. In Fig. 3(1) the downwind motion reduced velocity at the blade tip, blade root, and hub region (compared to Fig. 3(0)) can be identified and the wake starts to expand further in vertical direction from the blade tip. During the upwind move in Fig. 3(2), (3), and (4), regions with very low and even negative velocities are enlarging on the suction side of the blade and behind the hub.

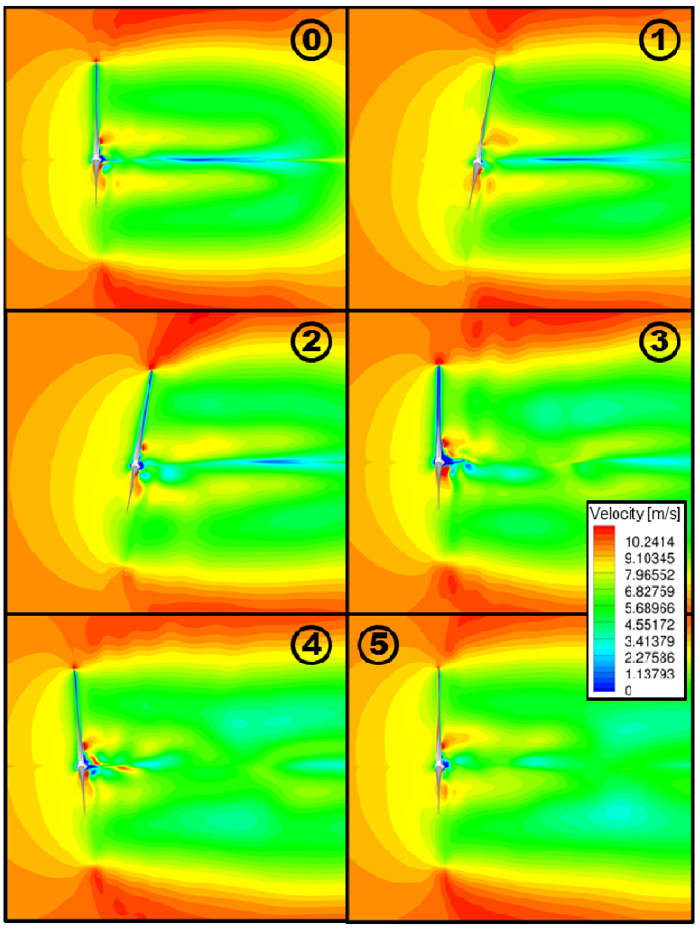

Fig. 3. Axial velocity contours of rotor and near wake

Although single vortices cannot be identified due to the coarse background grid used in this preliminary study (the vortices dissipate and are "blurred"), the global effects from vortices being shed from the blade tips and the inner blade region are visible. These include the further expansion of the wake and the "rippled" boundary region at the top of the wake in Fig. 3(3) and (4). When comparing the beginning (Fig. 3(0)) and end (Fig. 3(5)) of the motion, a changed flow field with a larger lower-velocity region in front of the rotor can be identified, as well as the expansion of the wake in the vertical and horizontal directions and the disappearance of the hub-shadow region in Fig. 3(0). The velocity at the tip also decreased. Because the single vortices are not resolved with the used-mesh resolution, the vortex ring state, as described by Sebastian and Lackner (2010; 2011), cannot be demonstrated with this study.

HYDRODYNAMICS

Morison Equation Limitations

Current integrated wind turbine design codes model the loads on fixed-bottom offshore structures by applying Morison's equation. Morison's equation is valid for slender cylinders and is a function of the diameter of the cylinder, fluid particle velocity and acceleration, and the hydrodynamic drag and inertia coefficients C_D and C_M. The drag and inertia coefficients are functions of Reynold's number, Keulegan-Carpenter number, and surface roughness, as well as a number of other factors. To calculate the applied hydrodynamic loads acting over the length of the structure, the cylinder can be divided into a number of elements in a manner similar to that of BEM theory. The total applied load is determined by integrating the loads acting on each element. The relative form of Morison's equation (Eq. 1) accounts for the relative motion between the cylinder and the fluid, and includes added mass effects from the movement of the water.

$$\mathbf{F}=\int_{-l_s/2}^{l_s/2}\left(\rho\frac{\pi D^2}{4}\frac{\partial v_n}{\partial t}+C_a\rho\frac{\pi D^2}{4}\frac{\partial u_{rn}}{\partial t}+C_d\rho\frac{D}{2}|u_{rn}|u_{rn}\right)ds, \quad (1)$$

Östergaard and Schellin (1987) describe this variation for slender hydrodynamic transparent cylindrical structures with arbitrary orientation to the current of the surrounding fluid. In Eq. 1 v denotes the fluid's velocity vector, u_r is the relative velocity vector between structure and fluid, and index n is the normal direction to the segment. The drag and inertia coefficients C_d and C_a are chosen empirically.

When it comes to modeling floating support structures, however, Morison's equation also has disadvantages. For support structures with a small diameter relative to the wavelength of the incident waves—that is, when the member diameter is less than 0.2 times the wavelength—diffraction effects can be neglected. This comes from G.I. Taylor's long-wavelength approximation. It states that, for surface-piercing bodies with a small diameter relative to the wavelength, the wave potential can be assumed to be constant across the body and calculations therefore can be performed at the centre of the body. Morison's equation uses this approximation to simplify the diffraction problem. When the submerged body has a diameter large enough for the waves to be disturbed by the presence of the structure, however, wave diffraction effects must be included for correct determination of the local pressure force and global wave loads. This often is the case for floating platforms, in particular for those stabilized by buoyancy, which means that Morison's equation cannot be used.

Morison's equation also assumes that viscous drag dominates the drag loading, and that wave radiation damping therefore can be ignored. This assumption is valid for slender structures and when the motions of the support structure are very small, which is usually the case for fixed-bottom support structures with soft-stiff characteristics. For floating platforms with sizeable volume and with low-frequency rigid modes, however, the support structure can experience significant movement, which means that wave radiation forces should be taken into account.

Morison's equation is used only for axisymmetric cylindrical structures, therefore it does not take account of any added mass-induced coupling between hydrodynamic force and support structure acceleration in all degrees of freedom present in nonaxisymmetric structures. Morison's equation also neglects hydrostatic restoring forces; however, additional terms can be added to account for this.

Linear Hydrodynamics Limitations

The most complete linear method of accounting for the different sources of hydrodynamic loading is to divide them into separate problems and solve them independently. To divide the hydrodynamics problem in this manner, the assumption of hydrodynamic linearity is required. This assumption also necessitates the use of linear Airy wave theory for the calculation of wave-particle kinematics. This approach is most commonly used in the oil and gas industry.

There are a number of important limitations introduced with the assumption of linearity. The use of Airy wave theory means that steep-sided or breaking waves found in shallow water cannot be modeled together with the resulting slap and slam loading. Additionally, the potential-flow theories used in a number of floating wind turbine design tools to calculate hydrodynamic loads were developed for stationary bodies, and only are valid when the support structure translational motion is small relative to the wavelength and the rotational motion is less than the wave steepness. Many floating configurations, however, experience large translational displacements relative to the length of the platform, for instance catenary moored systems where there is low resistance to surge and sway. This means that these potential-flow theories no longer are valid.

Second-Order Hydrodynamics

Linear wave theories also do not take into account second-order or higher order hydrodynamic effects, which are necessary for the analysis of platforms which are subject to steep-sided or very large waves. Second-order hydrodynamic loads are proportional to the square of the wave amplitude, and have frequencies equal to both the sum and the difference of the multiple-incident wave frequencies of an irregular sea state. This means that, although the natural frequencies of the structure are designed to be outside the wave energy spectrum, the second-order forces can excite these frequencies. Therefore, despite the forces normally being small in magnitude, the resonant effect can be important. The three components of second-order hydrodynamic forces for the diffraction problem are described below.

Mean drift forces. These forces result in a mean offset of the body relative to its undisplaced position, and typically are an order of magnitude less than first-order wave excitation forces. The mean drift force is a combination of second-order hydrodynamic pressure due to first-order waves and the interaction between first-order motion and the first-order wave field. The viscous drag could add to this force significantly when a current is present. The mooring-line tension often is related non-linearly to platform displacement, therefore the mean drift forces can have an important effect.

Slowly varying drift forces. These forces persist much longer than the main wave energy spectrum but still are within the range of horizontal platform motion. They result from non-linear interactions between multiple waves having different frequencies. Again, the forces resulting from slowly varying drift generally are small as compared to forces at the wave frequency, but they can cause large displacements in moored floating wind turbines which can in turn lead to high loads in the mooring lines. These forces also can excite the large amplitude resonant translational motion of the floating platform.

Sum frequency forces. These forces have a frequency which is higher than the wave frequency and also generally are small in amplitude. They arise from the same source as low-frequency drift forces, that is, from interactions between multiple waves of varying frequency. The contribution from these forces can be particularly important when analyzing "ringing" behavior for floating wind turbine configurations such as TLP concepts, which typically have high natural frequencies in heave, roll, and pitch. They also potentially can excite vibration modes of the supported wind turbine.

A recent study performed by Lucas (2011) on the comparison of first- and second-order hydrodynamics confirms the importance of second-order effects for an FOWT. The detailed comparison between the solution of first-order (linear) and second-order potential flow hydrodynamic models has been performed with regard to the characterization of the wave-induced loading and motion response under both regular and irregular waves. More specifically, the hydrodynamic quantities compared were the first- and second-order excitation force and the first- and second-order response amplitude operator (RAO) for unconstrained motions, analyzed for three distinct monochromatic waves and three unidirectional Pierson-Moskowitz spectra.

Two FOWT concepts featuring a 5-MW wind turbine were considered, the OC3-Hywind spar-buoy (Jonkman and Musial, 2010) and a semi-submersible platform with geometric dimensions similar to Principle Power's WindFloat platform concept, which comprises three equidistant columns and a wind turbine centered in one of the columns. For the WindFloat, hexagonal water-entrapment plates are installed at the bottom of each column to provide high heave added-mass and viscous damping to decrease the motions in this mode. The commercial software WAMIT, which solves the hydrodynamic problem in the frequency domain, was used to compute both the first-order (linear) and second-order (weakly nonlinear) hydrodynamic loads and unrestrained motions for the two structures considered in this study. Second-order excitations forces in Lucas' analysis are obtained as the sum of the force quadratic transfer functions (QTF) in the sum-frequency and difference-frequency. For a monochromatic wave, the second-order excitation force is expressed as the sum of only two components of the force QTF in the double and zero frequency. Note that, for the computation of the second-order hydrodynamic quantities, in addition to the platform itself, the free-surface also must be discretized.

In summary, for the OC3-Hywind, the second-order excitation force in stochastic waves is important in surge and pitch modes, although it still is smaller than the first-order excitation force. For the semi-submersible platform, the second-order excitation force is dominant over the first order for less-steep waves and lower significant wave heights for all modes except heave. The importance of the second-order excitation force decreases for the steeper waves and higher significant wave heights. The study shows that second-order effects prove to be more important for the semi-submersible concept than the OC3-Hywind, whose geometry is more hydrodynamically transparent. A detailed description of the investigation and results are given in Lucas (2011).

Even with second-order hydrodynamic terms included, however, the hydrodynamic theory might not completely apply to floating wind turbine platforms. This is because the theory was derived for use in the offshore oil and gas industry in which floating platforms typically have much smaller displacements than what is conventional for floating wind turbines.

Nonlinear hydrodynamic theories. Ongoing research at the University of Hamburg is the proposal of a nonlinear seakeeping simulation technique by using a Rankine-Airy panel method (Söding, 2010). The base flows from which the flow around the moving body is superimposed are not only source (and possibly vortex) flows, but also are Airy waves. Nonlinear boundary conditions at the free surface (constant pressure, no flux through the surface) are satisfied numerically in each time step by superimposing Airy waves of different wave numbers and propagation directions. The amplitudes and phase angles of the Airy waves are not constant over time, but must be computed from evolution equations. These are derived from the kinematic and dynamic free-surface conditions. The method is suitable for arbitrary geometries, and can handle most of the aforementioned nonlinear effects in steep waves. This method can be applied effectively for the estimation of extreme behavior of floating offshore wind turbines in survival conditions.

Vortex-Induced Vibrations

Another effect currently not accounted for in hydrodynamic analysis for floating wind turbines is vortex-induced vibration (VIV). This effect is caused by steady currents or by the velocities associated with long-period waves. It refers to the dynamic loading that occurs as a result of fluctuations in pressure due to the motion of vortices in the wake of a body. If the frequency of excitation is near a natural frequency of the structure, then the interaction between the flow and the motion of the structure can cause the two frequencies to resonate. This can result in large amplitudes of oscillation. The forces due to vortex shedding are complex, and predictions of loading and response are not well understood. The frequencies at which oscillations can occur, however, can be predicted with more confidence. Vortex-induced vibrations generally are not seen in conventional fixed-bottom offshore support structures, but are more likely to be experienced in mooring lines and can be critical for the stability of some designs.

MOORING-LINE DYNAMICS

Overview

Floating offshore wind turbine structures are held in position by means of mooring systems which, depending on the type of the structure and the water depth where it is to be moored, can have different levels of complexity. For floating wind turbine applications, a general distinction must be made between slack catenary, taut catenary, and taut tension-leg mooring systems. In slack catenary designs, the lower part of the line often rests on the seabed, which adds more complexity to the system. In the oil and gas industry, large floating drilling platforms are restored by up to 20 mooring lines with different geometrical and material properties. The lines consist of a combination of chains and cables made of natural or synthetic fibers (e.g., polyester, aramid, polyamide, polypropylene fibers). Submerged buoyancy tanks located along the mooring lines also are common. Such complex mooring solutions likely will be implemented and specially adapted for future floating wind turbines as well, thus requiring the codes to have adequate capabilities.

In addition to station-keeping, the mooring system also provides stability. For some platform designs, such as the tension-leg platform (TLP), the mooring system is the main contributor to the system's stability—meaning that a failure in this component would likely cause the destruction of the complete system. The mooring system of floating wind turbine platforms therefore is one of the most important components for the stability and the dynamic behavior of floating offshore wind turbines, making appropriate modeling of the mooring system highly critical during the design process.

The central issue with regard to mooring-line behavior is whether it is acceptable to neglect the dynamic effects of mooring lines for floating wind turbines. For shallow mooring systems the total mass of the lines is negligible and the motion is small. Therefore, even though the drag force of the lines through the fluid still might be significant, it generally is accepted that dynamics can be neglected. For deeper-water configurations, however, mooring-line dynamics become increasingly important. The conclusion drawn from studies performed for oil platforms and vessels was that line dynamic analysis should be conducted when the wave frequency response of the vessel is large, when the water depth exceeds 150 m, or when the mooring line includes large drag elements such as chain moorings.

To date, however, no studies have been dedicated to the dynamics of mooring lines for floating wind turbines. It is proposed that a study be performed specifically for floating wind turbines. It should investigate which aspects of mooring-line behavior are important, and determine the depth at which dynamic mooring-line effects become non-negligible for floating wind turbines, and which types of mooring system have the most dynamic effect on floating wind turbine platforms (and therefore must be designed using a full dynamic analysis). This study could incorporate a comparison between the different methods for calculating mooring-line tension forces for floating wind turbines.

The restoring forces on a floating wind turbine platform due to mooring lines can be approximated using force-displacement or quasi-static methods. These methods, however, do not account for mooring-line dynamic effects, such as line inertia, the drag of the line through fluid, and vortex shedding. For some configurations, these effects can have a significant contribution to the overall response of the system. A number of programs exist which can model the dynamics of mooring lines. These programs, however, generally do not allow for the detailed modeling of an integrated wind turbine system, including aeroelastic and aerodynamic forces, which make a significant contribution to the response of the whole system. Consequently, it is difficult to simultaneously account for the dynamic response of the mooring lines and the wind turbine in a single coupled analysis.

One approach is to use a dynamic line analysis code to derive a force-displacement relationship and apply this as a non-linear spring at the fairlead position. This method would give similar results as a quasi-static implementation. Another approach is to couple together dedicated mooring-line and wind turbine analysis codes. An attempt has been made by Jonkman and others to couple the dynamic mooring-line system LINES of SML with the aeroelastic wind turbine codes FAST and ADAMS. This attempt was abandoned after it was found that LINES encountered numerical instabilities when modeling the slack catenary mooring lines of interest. The most fruitful attempt to date is the coupling between offshore floating structures code SIMO/RIFLEX and the multibody wind turbine code HAWC2, described by Cordle and Jonkman (2011). This approach still is limited, however, in that the floating wind turbine cannot be modeled as a single integrated dynamic structure. The two problems must be solved in separate programs and information exchanged between the programs at a single interface point. This interface was known to be quite unstable numerically.

An approach for modeling mooring lines within integrated wind turbine design codes is to divide the mooring line into rigid (or flexible, modal reduced) multibody elements connected by spring-damper elements, originally described by Kreuzer and Wilke (2002) for oil platforms. The line-seabed interaction is modeled with a coulombic friction element including spring and hysteresis characteristics as a function of the translational forces. This MBS approach currently is being investigated by Matha and others, in which a multi-purpose commercial multibody code (SIMPACK, 2010; Matha, Hauptmann, Hecquet and Kühn, 2010) is extended to model offshore floating wind turbines. An originally implemented quasi-static mooring line model (within NREL's HydroDyn module) was replaced by an MBS-based model. With this approach, no interface between separate programs is necessary because the turbine's structure and mooring lines are modeled within one code using the same mathematical MBS formulation. This MBS formulation is numerically stable, integrated, and allows for a simple implementation of line-seabed interaction required for catenary systems. Following this MBS approach, the mooring system of the OC3-Hywind spar buoy in 320-meter deep water is modeled. The topology of the model of one mooring line is presented in Fig. 4.

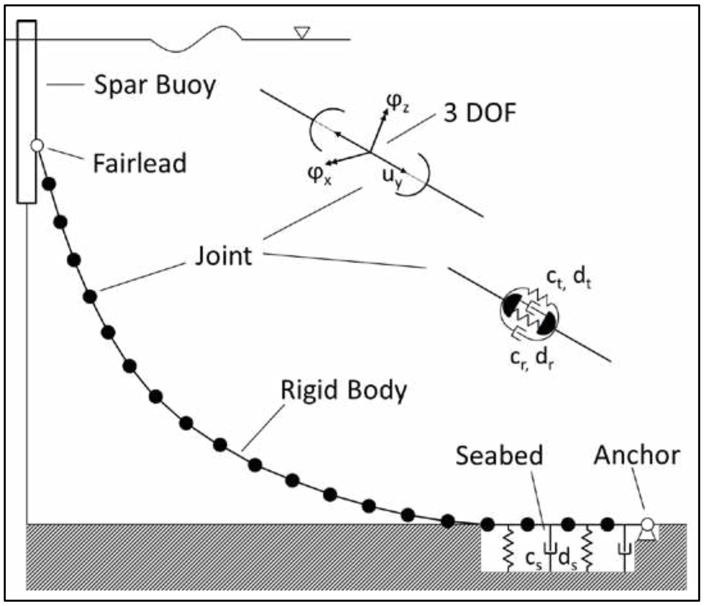

Fig. 4. Topology of MBS mooring-line model

Each of the spar buoy's three mooring lines is discretized into separate rigid bodies. Every single body is modeled as a cylindrical structure and has the gross properties of the particular part of the mooring line it represents, allowing modeling of different sectional mooring line configurations. The segments are connected by spring-damper elements to simulate the extensional (c_t) and rotational (c_r) stiffness and the accordant damping (d_t, d_r). Although this is a significant simplification of the structural arrangement of the line's fibers, it is necessary to reduce the model's complexity and simulation time. If using chain mooring lines, then this stiffness and damping could be neglected.

The degrees of freedom (DOF) relative to a prior body are reduced to a minimum of one translational (u_y) and two rotational (φ_x, φ_z) directions: To enable the simulation of the line's elongation, the DOF in cylinder's longitudinal direction must be maintained. The transverse rotational movements also must be enabled. The line's twist DOF was eliminated because there is no significant structural mooring-line torsion expected, and torsion only introduces minor hydrodynamic loads.
The very complex behavior of the seabed is reduced to a unilateral spring-damper model with high stiffness (c_s) and damping (d_s) to represent a rigid floor. The lateral friction is modeled by a coulombic element with an empiric friction coefficient.

The hydrodynamic effects on the mooring line are represented by a variation of the Morison Equation (Eq. 1), considering hydrodynamic drag and inertia, but neglecting effects based on dissipative flow, such as vortex induced vibrations.

Investigations of the discretization show only small effects on the results when increasing the number of elements beyond a certain number of elements. This model-specific number must be identified by a sensitivity analysis. Following this procedure enables the determination of a moderate discretization and limited simulation time, and retains the accuracy of the results.

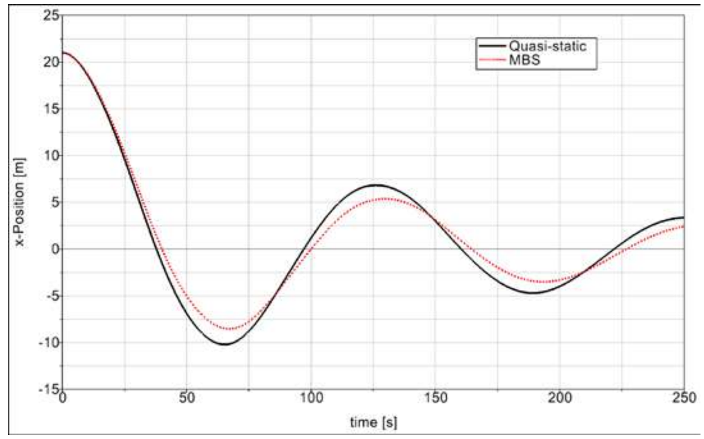

Fig. 5.Time series of OC3-Hywind platform translation in surge direction for OC3 DLC 1.4 surge

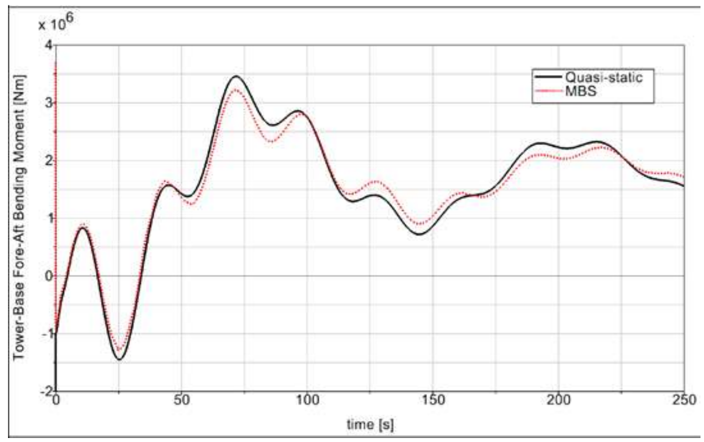

Fig. 6. Time series of OC3-Hywind tower-base fore-aft bending moment for OC3 DLC 1.4 surge

Comparisons of the MBS to quasi-static model in Fig. 5 for the platform's surge motion show differences caused by the non-linear additional hydrodynamic damping of the MBS model. Here, the platform initially is displaced 21 m in surge direction in still water (setup equals OC3 DLC 1.4 Surge) and then released, displaying the damped sine-shaped motion. The amplitudes of the motion are lower for the non-linear model and the periods are increased. Accordingly, similar differences in the fore-aft bending moment at the tower base can be identified in Fig. 6. Studies with stochastic sea states also show similar behavior.

CONCLUSIONS

State-of-the-art design tools for floating offshore wind turbines that can simulate such systems in an integrated way have already reached a viable level of sophistication. Nevertheless, the unique aerodynamics, hydrodynamics, and mooring-line dynamic effects occurring on floating offshore wind turbines due to large rotor and platform motions and the use of non-slender support structures require the further improvement of current modeling techniques to reach a greater confidence in simulated results. This paper presents the current challenges in simulation of FOWT. It examines both the important physical effects not yet addressed by current simulation codes, and the potential simulation developments and methodologies required to model these effects more accurately. Conclusions are drawn about the importance of the relevant effects, strengths, and weaknesses of the different methods and the development needed. Also analyzed is the applicability of the methods in FOWT design codes.

Regarding aerodynamics, the large low-frequency platform motions experienced by floating offshore wind turbines result in flow conditions that can be considerably more complex than those experienced by conventional onshore or fixed-bottom offshore wind turbines, and which are not captured by BEM. Significant pitch and surge motions lead to a change in the interaction between the rotor and wake. Dynamic stall and yawed inflow models also have increased importance for FOWTs. This paper discusses the limitations of BEM and the capabilities of computational fluid dynamics and potential flow methods (PFM) for FOWT simulation. Computational fluid dynamics results which show the complex flow conditions at the rotor occurring during a representative platform pitch motion are presented for the first time.

For modeling of non-slender floating platforms, hydrodynamic wave diffraction effects must be accounted for to correctly determine the local pressure force and global wave loads. Significant platform movements require inclusion of wave radiation forces; non-cylindrical elements demand modeling of added mass-induced coupling between hydrodynamic force and support structure acceleration. Their significance as well as the importance of second-order linear hydrodynamics and vortex-induced vibrations for FOWT simulation are discussed.

Different techniques for the representation of mooring-line dynamics, including quasi-static, look-up table, FEM and MBS methods, and their impact on global system loads are investigated. A MBS methodology is introduced and initial results are presented, indicating the importance of advanced non-linear mooring system models. Dedicated studies of the dynamics of mooring lines, including VIV effects, but specifically focusing on floating wind turbines are necessary.

Beyond theoretical and numerical analysis, validation of advanced modeling techniques with measurement data from full-scale prototypes is of great importance and is necessary to resolve the challenges discussed in this paper; especially when considering a potential new standard for the design of floating offshore wind turbines. The IEA Wind Task 30 Offshore Code Comparison Collaboration Continuation (OC4) project, which started in 2010, also will address some of these modeling challenges. In the future, advanced FOWT simulation tools will enable more reliable motion, load, and deflection predictions and ultimately will lead to improved designs to utilize the vast amount of wind resource located in deep waters.

ACKNOWLEDGEMENTS

The work presented here was funded partially by the Commission of the European Communities, Research Directorate-General within the scope of the Integrated Project "UpWind—Integrated Wind Turbine Design" (Project No. 019945 (SES6).

REFERENCES

Cordle, A, and Jonkman, J (2011). *State of the Art in Floating Wind Turbine Design Tools*. GL Garrad Hassan, ISOPE 2011.

DLR Braunschweig (2007). *FLOWer: Installation and User Manual of FLOWer MAIN version FLOWer 1-2007.1.*

International Electrotechnical Commission (2009). IEC 61400-3, *Wind Turbines—Part 3: Design Requirements for Offshore Wind Turbines*.

Jonkman, J (November 2007). *Dynamics Modeling and Loads Analysis of an Offshore Floating Wind Turbine*. NREL/TP-500-41958. Golden, CO.

Jonkman, J, and Matha, D (2011). "Dynamics of offshore floating wind turbines - analysis of three concepts," *NREL, Wind Energy Journal*, DOI: 10.1002/we.442 2011.

Jonkman, J, and Musial, W (2010). *Final Technical Report, IEA Wind Task 23, Subtask 2, Offshore Code Comparison Collaboration (OC3)*, International Energy Agency (IEA), IEA Wind Task 23, Offshore Wind Technology and Deployment.

Kreuzer, E, and Wilke, U (2002). "Mooring Systems—A Multibody Dynamic Approach," Technical University Hamburg-Harburg, *Multibody System Dynamics,* Vol 8.

Lucas, J (January 2011). *Comparison of First and Second-Order Hydrodynamic Results for Floating Offshore Wind Structures*, GL Garrad Hassan, Report Ref: 11594br02a.

Matha, D (2009). *Model Development and Loads Analysis of an Offshore Wind Turbine on a Tension Leg Platform, with a Comparison to Other Floating Turbine Concepts*. NREL/TP-500-45891. Golden, CO.

Matha, D, Hauptmann, S, Hecquet, T, and Kühn, M (2010). *Methodology and Results of Loads Analysis of Wind Turbines with Advanced Aeroelastic Multi-Body Simulation*, DEWEK, Bremen, Germany.

Meister, K, Lutz, T, and Krämer, E (2010). *Consideration of Unsteady Inflow Conditions in Wind Turbine CFD Simulations,* DEWEK, Bremen, Germany, 2010.

Östergaard and Schellin (1987). "Comparison of experimental and theoretical wave actions on floating compliant offshore structures," *Applied Ocean Research 9*, pp 192-213

Robertson, A, and Jonkman, J (2011). *Loads Analysis of Several Offshore Floating Wind Turbine Concepts*. NREL, ISOPE 2011.

Sebastian, T, and Lackner, M (2010). *A Comparison of First-Order Aerodynamic Analysis Methods for Floating Wind Turbines*, 48th AIAA Aerospace Sciences Meeting, Orlando, Florida, 2010.

Sebastian, T, and Lackner, M (2011). *Offshore Foating Wind Turbines - An Aerodynamic Perspective*, 49th AIAA Aerospace Sciences Meeting, Orlando, Florida, 2011.

SIMPACK (2010). "SIMPACK Reference Guide," *SIMDOC* v8.904.

Söding, H (2010). "Boundary Element Methods for Seakeeping," *Technical University Hamburg-Harburg, Duisburg, Contribution to the Lectures on Seakeeping and Sea Loads*.

Streiner, S, Hauptmann, S, Kühn, M, and Krämer, E (2008). *Coupled Fluid-Structure Simulations of a Wind Turbine Rotor*, University Stuttgart, DEWEK