Coupled fluid-structure interaction simulation of floating offshore wind turbines and waves: a large eddy simulation approach

Antoni Calderer, Xin Guo, Lian Shen and Fotis Sotiropoulos

St. Anthony Falls Laboratory, Department of Civil Engineering, Department of Mechanical Engineering, University of Minnesota, 2 Third Avenue SE, Minneapolis, MN 55414, USA
1 Corresponding author: fotis@umn.edu

Abstract. We develop a computational framework for simulating the coupled interaction of complex floating structures with large-scale ocean waves and atmospheric turbulent winds. The near-field approach features a partitioned fluid-structure interaction model (FSI) combining the curvilinear immersed boundary (CURVIB) method of Borazjani and Sotiropoulos (J. Comput. Phys. 2008) and the two-phase flow level set formulation of Kang and Sotiropoulos (Adv. in Water Res. 2012) and is capable of solving complex free-surface flows interacting non-linearly with complex real life floating structures. The near-field solver is coupled with a large-scale wave and wind model based on the two-fluid approach of Yang and Shen (J. Comput. Phys. 2011) which integrates a viscous Navier-Stokes solver with undulatory boundaries for the motion of the air and an efficient potential-flow based wave solver. The large-scale turbulent wind is incorporated from the far-field solver to the near-field solver by feeding into the latter inlet boundary conditions. The wave field is incorporated to the near-field solver by using the pressure-forcing method of Guo and Shen (J. Comput. Phys. 2009) which has been appropriately adapted to the level set method. The algorithm for coupling the two codes has been validated for a variety of wave cases including a broadband spectrum showing excellent agreement when compared to theoretical results. Finally, the capabilities of the numerical framework are demonstrated by carrying out large eddy simulation (LES) of a floating wind turbine interacting with realistic ocean wind and wave conditions.

1. Introduction

The high potential of floating wind turbines to capture part of the vast offshore wind energy resource has attracted increasing attention by the scientific community. High-fidelity numerical simulations can play a major role in the development of novel offshore wind technologies, and may be the only feasible way to tackle such a problem. However, the complexity of the problem poses a major challenge due to the need to resolve the coupled interaction of atmospheric turbulence and ocean waves, the arbitrary geometric complexity of floating structures, the inherent two-phase nature of such flows, and the dominant role of complex nonlinear phenomena such as turbulence and free surface effects.

Most numerical models in the literature for simulating floating wind turbines are based on oversimplified assumptions, such as assuming inviscid and irrotational flow, and accounting for the wave loading by solving the semi-empirical Morison’s equation [1–5]. These models, however, are not able to capture the structural response to unsteady flows and turbulence effects. As demonstrated by Sebastian and Lackner [6] the aerodynamics of floating wind turbines are far more complex than that of land based systems, and as a result, higher order models are
required for obtaining accurate solutions. The only approach that can in principle model the aerodynamics of offshore wind turbines with turbulence and viscous effects is by employing a Navier-Stokes solver with fluid-structure interaction (FSI).

Offshore phenomena generally occur at very large scales. An example is the formation of swells which require a domain in the scale of kilometers. When studying an offshore floating structure, which is typically in the scale of meters, the problem of large disparity of scales arises. The objective of this work is to present a powerful numerical framework capable of simulating the coupled interactions of a complete floating wind turbine with large-scale ocean waves and atmospheric turbulent winds. The approach couples an efficient large-scale model which is referred in this work to as the far-field flow solver and is suitable for simulating realistic ocean wave and wind conditions, with a high resolution near-field model capable of solving complex free-surface flows interacting non-linearly with arbitrarily complex real life floating structures. The large-scale wave and wind model is based on the two-fluid approach of Yang and Shen [7, 8] which integrates a viscous Navier-Stokes solver with undulatory boundaries for the motion of the air and an efficient potential-flow based wave solver. On the other hand, the near-field approach was recently develop by Calderer et al. [9] and features a partitioned FSI model combining the curvilinear immersed boundary (CURVIB) method of Borazjani and Sotiropoulos [10] and the two-phase flow level set formulation of Kang and Sotiropoulos [11].

While direct numerical simulation (DNS) of the flow around a floating turbine offers the highest accuracy, the large Reynolds number flow of offshore real life applications makes it computationally impractical. The method we propose, based on large-eddy simulation (LES), can simulate turbulent flow with high accuracy by only solving the large eddies of the flow and modeling the smallest scales. Since the computational cost is still very demanding, the method can be employed as a tool to develop and test physics-based, low-dimensional dynamic models. Also it can be employed to gain a better understanding of floating turbine dynamics and characterize the wake structure behind the turbine rotor.

2. The near-field flow solver

The near-field model solves the spatially-filtered incompressible Navier-Stokes equations using a two-phase flow level set formulation. The flow properties in this approach are variables adopting in each phase its corresponding value, and smoothly transitioning over a thin layer of thickness $2\epsilon$ across the interface as follows

$$\rho(\phi) = \rho_{\text{air}} + (\rho_{\text{water}} - \rho_{\text{air}}) h(\phi),$$

$$\mu(\phi) = \mu_{\text{air}} + (\mu_{\text{water}} - \mu_{\text{air}}) h(\phi),$$

where $\rho$ is the density, $\mu$ is the viscosity, $\phi$ is a signed distance function used for tracking the position of the interface, and $h(\phi)$ is the smoothed Heaviside function [12]. Then the governing equations in generalized curvilinear coordinates read as follows

$$J \frac{\partial U^i}{\partial t} = 0,$$

$$\frac{1}{J} \frac{\partial J U^j}{\partial t} = \xi_j^i \left( - \frac{\partial}{\partial \xi_j} (U^j u_i) + \frac{1}{\rho(\phi)} \frac{\partial}{\partial \xi_j} \left( \mu(\phi) \frac{\xi_k^j \xi_l^i}{J} \frac{\partial u_k}{\partial \xi_l} \right) - \frac{1}{\rho(\phi)} \frac{\partial}{\partial \xi_j} \left( \frac{\xi_k^j p}{J} \right) - \frac{1}{\rho(\phi)} \frac{\partial}{\partial \xi_j} \tau_{ij} + \delta_{ij} g + S_i \right),$$

where $\xi_j^i$ are the transformation metrics, $J$ is the Jacobian of the transformation, $u_i$ are the Cartesian velocity components, $U^i$ are the contravariant volume fluxes, $p$ is the pressure, $\tau_{ij}$ is the sub-grid stress (SGS) tensor, $\delta_{ij}$ is the Kronecker delta, $g$ is the gravity and $S_i$ is the forcing term for wave generation.
For simulating complex flows with large eddy simulation (LES), we use the dynamic Smagorinsky SGS model of [13] in combination with a wall modeling strategy. The motion of the air-water interface is modeled by solving the following level set equation

$$ \frac{1}{J} \frac{\partial \phi}{\partial t} + U_j \frac{\partial \phi}{\partial \xi^j} = 0 $$

(5)

A mass conserving re-initialization equation is then solved to ensure proper conservation of mass within the two fluids as extensively described in Kang and Sotiropoulos [11].

The momentum equations (4) are discretized using a second-order central differencing scheme for the diffusion and advective terms, except in regions such as the vicinity of the free surface interface where the third-order WENO scheme [14] is applied for the advective terms. The solution is advanced in time by using a second-order Crank-Nicholson scheme and the fractional step method. The level set equations (5) are discretized with a third-order WENO scheme in space, and second-order Runge-Kutta in time. The re-initialization step uses a second-order ENO scheme [15].

To simulate the dynamic motion of complex floating structures interacting with two-phase free surface flows we employ an extension of the FSI-CURVIB method of Borazjani and Sotiropoulos [10] integrated with the previously discussed level set formulation. The FSI method of [10] adopts a partitioned approach to couple the flow field with the dynamics of rigid bodies governed by the 6 degree of freedom (DoF) equations of motion (EoM).

3. The far-field flow solver
The large-scale wave-wind model is based on the two-fluid coupled approach of Yang and Shen [8], which employs a potential based wave solver of high-order spectral method for the water motion and a viscous solver with undulatory boundaries for the air motion. A brief description of the governing equations and numerical methods as well as the coupling algorithm is provided in this section.

3.1. High-order spectral method (HOS)
We solve the potential flow wave problem formulated in the form of Zakharov [16] by applying the HOS method of [17]. Kinematic and dynamic boundary conditions (BCs) can be written as functions of the free surface elevation $\eta$ and the velocity potential $\Phi$ as follows

$$ \nabla^2 \Phi = \frac{\partial^2 \Phi}{\partial x_i \partial x_i} = 0, $$

(6)

$$ \frac{\partial \eta}{\partial t} + \frac{\partial \eta}{\partial x_\alpha} \frac{\partial \Phi^s}{\partial x_\alpha} \left(1 + \frac{\partial \eta}{\partial x_\alpha} \frac{\partial \eta}{\partial x_\alpha}\right) \frac{\partial \Phi}{\partial x_3} \bigg|_{x_3=\eta} = 0, $$

(7)

$$ \frac{\partial \Phi^s}{\partial t} + g \eta + \frac{1}{2} \frac{\partial \Phi^s}{\partial x_\alpha} \frac{\partial \Phi^s}{\partial x_\alpha} \left(1 + \frac{\partial \eta}{\partial x_\alpha} \frac{\partial \eta}{\partial x_\alpha}\right) \left(\frac{\partial \Phi}{\partial x_3}\right)^2 \bigg|_{x_3=\eta} = -P_a, $$

(8)

where $i = 1, 2, 3$, $\alpha = 1, 2$, and $\Phi^s = \Phi|_{x_3=\eta}$ and $P_a$ are the velocity potential and air pressure at the water surface, respectively.

Periodic BCs are imposed on the horizontal directions, which allows the HOS method to use an efficient spatial discretization scheme based on a pseudo-spectral method. The equations (7) and (8) are advanced in time with the fourth-order Runge-Kutta (RK4) scheme.
3.2. LES method
For the air flow over water waves in the far-field model, we solve the following filtered incompressible Navier-Stokes equations governing the flow of a single fluid

\[
\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial (\tilde{u}_i \tilde{u}_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \tilde{P}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j},
\]

\[
\frac{\partial \tilde{u}_i}{\partial x_i} = 0.
\]

where \( \tilde{u}_i \) are the filtered velocity components, \( \tilde{P} \) is the filtered dynamic pressure, and \( \tau_{ij} \) is SGS stress tensor modeled by the scale-dependent Lagrangian dynamic model [18]. Note that the viscous term in equation (9) has been neglected due to the consideration of high Reynolds number and the negligible effect of viscosity at the resolved scale. A wall model is used to account for the viscous effects at the bottom free surface boundary. A boundary fitted grid is employed and adapts to the motion of the free-surface which is seen by the air domain as a undulatory boundary. The geometry and the velocity of the bottom boundary are prescribed from the HOS simulation. The air flow is lid driven by a shear stress at the top boundary and periodic BCs are considered in the horizontal directions.

For the spacial discretization in the horizontal directions, a Fourier-series-based pseudospectral method is used, while in the vertical direction a second-order finite difference scheme is used. A semi-implicit fractional-step method is applied to advance the governing equations (9) and (10).

3.3. LES-HOS coupling algorithm
The LES and HOS models are dynamically coupled by following an iterative procedure. First the HOS method is used to advance the wave to the next time step \( (n+1) \) under the forcing of air pressure \( P^n_a \) on wave surface, i.e., the free surface elevation \( \eta^{n+1} \) and surface velocity \( u^{n+1}_s \) which can then be imposed as Dirichlet BC at the bottom boundary of the LES model. The air flow can then be advanced to time step \( (n+1) \) by solving the described LES model, and a new value of surface pressure \( P^{n+1}_a \) is computed to continue the simulation.

4. Far-field/near-field coupling algorithm
The far-field wind-wave solver is loosely coupled with the near-field FSI solver. The wind flow from the far-field can be incorporated directly into the near-field solver by prescribing the instantaneous air velocity at the inlet boundary. The process for incorporating the wave field involves the following two steps: (1) extract the energy and phases of surface waves from the far-field model by performing a Fourier analysis, and (2) prescribe the resulting waves to the near field by applying the surface forcing method of Guo and Shen [19] which has been appropriately adapted for the present level set method.

4.1. Near-field wave generation method
Wave fields are generated in the near-field domain by applying a force on the free surface as proposed by [19]. A schematic description of the overall procedure is depicted in figure 1, showing how the waves propagate symmetrically to both sides of the source region and towards the lateral boundaries where they are dissipated by a sponge layer. To generate the following surface elevation

\[
\eta(x, y, t) = A \cos(k_x x + k_y y + \theta),
\]
where $A$ is the wave amplitude, $k_x$ and $k_y$ the components of the wave number vector, and $\theta$ is the wave phase, a forcing method is applied by adding a source term, equivalent to a nodal force, in the filtered momentum equations. To adapt the forcing method of [19] to the level set method the nodal force applied at the free surface has been smeared in a distance $2\epsilon_\phi$ along the normal direction and $2\epsilon_x$ along the $x$ direction. The so constructed forcing term reads as follow

$$S_i(x, y, t) = n_i(\phi) P_0 \delta(x, \epsilon_x) \delta(\phi, \epsilon_\phi) \sin(\omega t - k_y y - \theta),$$

(12)

where $n_i$ denotes the normal direction of the free surface, $\omega$ is the wave frequency, $P_0$ is a coefficient that depends on the wave and fluid characteristics, and $\delta$ is a distribution function defined as

$$\delta(x, \epsilon) = \frac{1}{2\epsilon} \left[ 1 + \cos \left( \frac{\pi x}{\epsilon} \right) \right].$$

(13)

Typical values adopted in this work for $\epsilon_x$ are in the order of half wavelength, and for $\epsilon_\phi$ between 3 and 6 grid sizes. By applying superposition principles, the above method can be applied to generate complex wave fields with multiple wave frequencies as it is demonstrated in the result section. The sponge layer method of Choi et al. [20] is applied at the boundaries to prevent wave reflexions.

5. Results

A thorough validation of the FSI near-field solver was already demonstrated in the work of Calderer et al. [9] showing excellent agreement with other numerical and experimental studies. In this section we present one representative case from [9] to illustrate the ability of the near-field FSI model to simulate the dynamics of floating structures. We subsequently focus on presenting a number of test cases to validate the proposed algorithm for coupling the far-field/near-field flow solvers and illustrate the potential of the computational framework to simulate floating turbine dynamics.

5.1. Near field model FSI validation: Free roll decay test of a rectangular barge

We consider in this section the free roll decay test of a rectangular barge. The configuration, illustrated in figure 2a, is identical to that studied experimentally by Jung et al. [21]. The barge is 0.3m wide, 0.9m long and 0.2m tall, and is located in the center section of a 35m long rectangular channel. The width of the channel as well as the water depth is equal to 0.9m. The center of gravity of the barge coincides with the center of rotations which is aligned with the free surface water level. The barge is initially inclined with a 15 degree angle and starts oscillating freely until the motion is dampened by frictional and viscous effects. The rotational inertia is $I = 0.236kgm^2$.

A non-uniform grid is employed composed of an inner rectangular region enclosing the structure ($-0.8m \leq x \leq 0.8m$) in the horizontal direction and $-0.3m \leq z \leq 0.3m$ in the vertical direction) with constant grid spacing equal to 0.001m in both the vertical and horizontal
Figure 2: (a) Schematic description of the rectangular barge configuration. (b) Comparison between the computed angle of inclination of the barge and the experimental data of Jung et al. [21].

Figure 3: Snapshots of the rectangular barge simulation with the corresponding free surface and out-of-plane vorticity contours.

directions. Outside of this inner region the grid is gradually coarsened away from the structure. The overall size of the mesh is approximately 10.5 million, with $1300 \times 740 \times 11$ nodes in the horizontal, vertical, and spanwise directions, respectively. The time step for the simulation is $0.0005\,s$.

To account for the frictional effect inherent of the experimental result, we introduced an artificial damping coefficient $C = 0.275$ in the equation of motion (see Calderer et al [9] for more details and an extensive grid refinement study). The angular response of the structure is compared with the experimental results of [21] in figure 2b and very good agreement is observed. In figure 3 we show the flow patterns at different times of the barge oscillation process including the free surface configuration and out-of-plane vorticity contours. Complex vortex formation and shedding phenomena are observed both at the edges of the rectangular structure and at the crest of the induced waves.

In summary, the presented results along with the more detailed comparisons in [9]
demonstrate the validity of the FSI model in predicting the structural motion of a floating structure and capture of the complex flow phenomena resulting from the interaction of the structure with the free surface interface.

5.2. Forcing method validation case: directional waves

In this case we validate the forcing method for wave generation by generating a linear directional wave field in a 3-dimensional basin of constant depth. The wave amplitude we consider is $A = 0.01 m$, the wavelength $L = 1.2 m$, and the wave direction $\beta = 30 \text{deg}$. The domain length is $20 m$ ($-10 m \leq x \leq 10 m$) in the longitudinal direction and $25 m$ ($-10 m \leq y \leq 15 m$) in the span-wise direction, the depth of water and air is $2 m$ and $1 m$, respectively. A non-uniform grid of size $186 \times 125 \times 171$ in the $x, y$ and $z$ directions, respectively, is employed consisting on an inner rectangular region with uniform grid spacing and an outer region within which the mesh is gradually stretched towards the boundaries. The inner region ($-5 m \leq x \leq 5 m, -10 m \leq y \leq 10 m, -0.1 m \leq z \leq 0.1 m$) which contains the source region and part of the propagated waves, has a constant grid spacing of $0.08 m, 0.143 m$ and $0.005 m$, in the $x, y$ and $z$ directions, respectively. The source region is centered on $x = 0$ with thickness $\epsilon_x = L/2 = 0.6 m$, $\epsilon_\phi = 0.06 m$ and spans the entire domain along the $y$ direction. The time step used is of $0.00125 s$, the interface thickness is $0.02 m$, the gravity is $g = -10 m/s^2$ and the sponge layer method with length $1.2 m$ is applied at the horizontal boundaries.

The free surface elevation and its comparison with the analytical solution is presented in figure 4 showing the high accuracy of the surface forcing method in generating directional waves. The only exception is in the source region where the simulated results are not expected to follow the analytical free surface pattern.

![Figure 4: Comparison between the simulation and theoretical surface elevation of directional waves. Computed and analytical surface elevation along cross planes (left) at (a) $Y = 0.0 m$ and (b) $X = 2.0 m$, and top view of the computed surface elevation (right), at time $t = 15 s$. The cross planes of left figures are marked in the top view of the computed surface elevation in right figures.](image-url)
5.3. Far-field/near-field coupling case: broadband wave spectrum

This case was designed to (1) validate the forcing method in the generation of waves with broadband wave spectrum, and (2) validate the algorithm that extracts the wave field from the far-field solver and incorporates it to the near field solver. A broadband wave spectrum obtained during the Joint North Sea Wave Project (JONSWAP) is used as initial condition of the far-field solver. A Fast Fourier Transform of the surface elevation is applied to extract the wave frequencies and amplitudes, which are then incorporated to the near field solver with the surface forcing method (section 4). The near-field computational domain is $19m$ long ($-7 \leq x \leq 12$), $10m$ wide ($-5 \leq y \leq 5$) and $0.5m$ depth. The grid of size $275 \times 112 \times 92$ is constructed with the same two-region grid structure discussed in the previous test case. The inner region ($-2.5m \leq x \leq 2.5m, -2.5m \leq y \leq 2.5m, -0.05m \leq z \leq 0.05m$) has a constant grid spacing of $0.05m$, $0.08m$ and $0.005$ in the $x$, $y$ and $z$ direction, respectively. A sponge layer of length $3m$ is applied at the inlet/outlet boundaries, and of $2m$ for the lateral boundaries, the time step is $0.002s$ and the gravity $g = -9.81m/s^2$.

The near-field surface elevation for this case is presented in figure 5 and its comparison with the expected analytical solution demonstrates the accuracy of the presented algorithm to simulate realistic ocean wave fields.

![Figure 5: Far-field/near-field coupling test case. Computed (color contour) and analytical (contour lines) free surface elevation of the broadband wave spectrum incorporated from the far-field to the near-field solver.](image)

5.4. FSI simulation of a floating wind turbine

In this final case we seek to demonstrate the capabilities of the presented numerical framework by simulating a laboratory scale floating wind turbine and its interaction with the corresponding scaled down conditions representative of an offshore environment. Therefore, we have fed the near-field solver with a fully developed turbulent wind field from a pre-computed LES and a broadband wave field with JONSWAP wave spectrum as initial condition.

The floating turbine is composed of a cylindrical platform of radius $R = 0.28m$, height $H = 0.15m$ and draft $0.08m$. The turbine rotor diameter is $0.6m$ and the hub height is located $0.99m$ above the free surface level. We account for the effect of the rotor by employing the actuator disk model implemented as in Yang et al. [22]. Since the scope of the present case is to illustrate some of the capabilities of the approach, we simplified the problem by neglecting the effects of the mooring system, and only allowing the structural motion in two DoF, i.e., heave and pitch, which are known to be among the most relevant DoF in floating turbines.

The computational domain is a 3-dimensional channel of length $16m$ ($-6 \leq x \leq 10$), width $8.5m$ ($-4.25 \leq y \leq 4.25$), depth $0.5m$ and air height $2m$. The non-uniform grid we employed
for this case is of size $435 \times 97 \times 170$ and has two inner regions with constant grid spacing, one containing the turbine rotor and another containing the platform. The former has a vertical spacing of 0.02$m$ and the latter of 0.01$m$. The grid spacing in the x and y directions is the same for the two regions and equal to 0.25$m$ and 0.05$m$, respectively. The gravity is set to $g = -9.81m/s^2$ and the time step of the simulation is 0.0023s. The sponge layers at the inlet and outlet boundaries are 2.5$m$ long, and 2$m$ long at the lateral boundaries.

A snapshot of the turbine configuration, the free surface elevation, and contours of stream-wise velocity is presented in figure 6. The same figure also shows the time history of heave and pitch of the turbine illustrating the structural response complexity resulting from the interaction of the platform with the wave field and turbulent wind. As one would expect, the pitch inclination is slightly shifted towards positive angles (clockwise) which is caused by the effect of the wind.

![Figure 6: Contours of the wind stream-wise velocity around a floating wind turbine (left). Response in pitch and heave of the floating structure (right).](image)

### 6. Summary and future work

The objective of the present work was to develop a computational framework that can simulate real-life complex floating structures and its interaction with realistic ocean wave and wind fields. The validity and performance of the proposed far-field/near-field coupling algorithm based on the surface forcing method of Guo and Shen [19] were systematically verified by comparing two wave cases of increasing complexity with theoretical solutions derived from linear wave theory. To demonstrate the potential of the method we apply it to carry FSI simulation of a laboratory scale floating wind turbine with 2 DoF within a broadband wave field and a developed large-scale turbulent air flow.

As a future work, we will further validate the present numerical framework with a set of experimental studies currently under development in the St. Anthony Falls Laboratory. In particular, the structural dynamics of a floating turbine exposed to different wave cases will be thoroughly analyzed. We will then apply the model to simulate a real-life offshore floating wind turbine including all the 6 DoF motions as well as the effect of the mooring system.

### Acknowledgments

This work has been supported by the US Department of Energy (DE-EE 0005482) and the University of Minnesota Initiative for Renewable Energy and the Environment. Computational resources were provided by the Minnesota Supercomputing Institute.
References

[1] M. J. Kuhn, *Dynamics and design optimisation of offshore wind energy conversion systems*. PhD thesis, Wind Energy Research Institute, Delft University of Technology, Delft, The Netherlands, June 2001.

[2] P. W. Cheng, *A reliability based design methodology for extreme responses of offshore wind turbines*. PhD thesis, Wind Energy Research Institute, Delft University of Technology, Delft, The Netherlands, Oct. 2002.

[3] H. Veldkamp and J. Van Der Tempel, “Influence of wave modelling on the prediction of fatigue for offshore wind turbines,” *Wind Energy*, vol. 8, no. 1, pp. 49–65, 2005.

[4] J. M. Jonkman, “Dynamics of offshore floating wind turbines: model development and verification,” *Wind Energy*, vol. 12, no. 5, pp. 459–492, 2009.

[5] S. Gueydon and W. Xu, “Floating wind turbine motion assessment,” pp. 1–10, 2011.

[6] T. Sebastian and M. A. Lackner, “Characterization of the unsteady aerodynamics of offshore floating wind turbines,” *Wind Energy*, 2012.

[7] D. Yang and L. Shen, “Simulation of viscous flows with undulatory boundaries. part i: Basic solver,” *Journal of Computational Physics*, vol. 230, pp. 5488–5509, June 2011.

[8] D. Yang and L. Shen, “Simulation of viscous flows with undulatory boundaries: Part II. coupling with other solvers for two-fluid computations,” *Journal of Computational Physics*, vol. 230, pp. 5510–5531, June 2011.

[9] A. Calderer, S. Kang, and F. Sotiropoulos, “Level set immersed boundary method for coupled simulation of air/water interaction with complex floating structures,” *Journal of Computational Physics, Under Review*, 2014.

[10] I. Borazjani, L. Ge, and F. Sotiropoulos, “Curvilinear immersed boundary method for simulating fluid structure interaction with complex 3D rigid bodies,” *Journal of Computational Physics*, vol. 227, pp. 7587–7620, Aug. 2008.

[11] S. Kang and F. Sotiropoulos, “Numerical modeling of 3D turbulent free surface flow in natural waterways,” *Advances in Water Resources*, vol. 40, pp. 23–36, May 2012.

[12] S. J. Osher and R. P. Fedkiw, *Level Set Methods and Dynamic Implicit Surfaces*. Springer, 2003 ed., 2002.

[13] M. Germano, U. Piomelli, P. Moin, and W. H. Cabot, “A dynamic subgrid-scale eddy viscosity model,” *Physics of Fluids A: Fluid Dynamics*, vol. 3, no. 7, pp. 1760–1765, 1991.

[14] G.-S. Jiang and C.-W. Shu, “Efficient implementation of weighted ENO schemes,” *Journal of Computational Physics*, vol. 126, pp. 202–228, June 1996.

[15] M. Sussman, E. Fatemi, P. Smereka, and S. Osher, “An improved level set method for incompressible two-phase flows,” *Computers & Fluids*, vol. 27, pp. 663–680, June 1998.

[16] V. E. Zakharov, “Stability of periodic waves of finite amplitude on the surface of a deep fluid,” *Journal of Applied Mechanics and Technical Physics*, vol. 9, no. 2, pp. 190–194, 1968.

[17] D. G. Dommermuth and D. K. Yue, “A high-order spectral method for the study of nonlinear gravity waves,” *Journal of Fluid Mechanics*, vol. 184, pp. 267–288, 1987.

[18] E. Bou-Zeid, C. Meneveau, and M. Parlange, “A scale-dependent Lagrangian dynamic model for large eddy simulation of complex turbulent flows,” *Physics of Fluids*, vol. 17, no. 2, 2005.

[19] X. Guo and L. Shen, “On the generation and maintenance of waves and turbulence in simulations of free-surface turbulence,” *Journal of Computational Physics*, vol. 228, pp. 7313–7332, Oct. 2009.

[20] J. Choi and S. B. Yoon, “Numerical simulations using momentum source wave-maker applied to RANS equation model,” *Coastal Engineering*, vol. 56, pp. 1043–1060, 2009.

[21] K. H. Jung, K. A. Chang, and H. J. Jo, “Viscous effect on the roll motion of a rectangular structure,” *Journal of engineering mechanics*, vol. 132, no. 2, pp. 190–200, 2006.

[22] X. Yang, S. Kang, and F. Sotiropoulos, “Computational study and modeling of turbine spacing effects in infinite aligned wind farms,” *Physics of Fluids*, vol. 24, no. 11, 2012.