Development and Validation of a Tightly Coupled CFD/6-DOF Solver for Simulating Floating Offshore Wind Turbine Platforms

Alexander J. Dunbar - Corresponding Author

Department of Mechanical and Nuclear Engineering The Pennsylvania State University University Park

Contact Information: Room 17 Reber Building, University Park, PA 16802

ajd315@psu.edu

Brent A. Craven

Department of Mechanical and Nuclear Engineering The Pennsylvania State University University Park

Eric G. Paterson - Corresponding Author

Aerospace and Ocean Engineering Virginia Tech

Contact Information: Randolph Hall, RM 215, 460 Old Turner Street, Blacksburg, VA 24061

(540) 231-2314 egp@vt.edu

Abstract

Simulations of offshore floating wind turbine platform dynamics typically utilize engineering tools that include simplified modeling assumptions. In this study, an open-source CFD/6-DOF solver is developed using OpenFOAM for high-fidelity simulation of offshore floating wind turbine platforms. The solver tightly couples the fluid and 6-DOF equations of motion using subiterations and dynamic relaxation to eliminate the artificial added mass instability. Validation of the tightly coupled CFD/6-DOF solver is carried out on a benchmark case of the free decay of a heaving cylinder. The solver is then used in simulations of the DeepCwind semisubmersible platform and compared with FAST, a NREL engineering tool. The CFD/6-DOF solver and FAST are compared in four cases including both rotational and translational motion. Overall, the results demonstrate that the tightly coupled solver compares well with FAST. The tightly coupled CFD/6-DOF solver represents an advance in modeling of offshore wind turbine platform dynamics using open-source software that may be used for wind turbine research, design, and analysis.

1. Introduction

Due to the limited deployment range of fixed tower offshore wind turbines, recent focus has shifted to floating offshore platforms. Current methods for modeling the dynamics of floating offshore wind turbine systems rely primarily on simplified assumptions using coefficient-based models [7, 19, 38, 39]. However, due to the complexity of integrating a wind turbine with a floating platform, higher fidelity methods of analysis are desired.

In recent efforts to assess and improve the current state of the art in modeling floating offshore wind turbines, the International Energy Agency has sponsored the Offshore Code Comparison Collaboration (OC3) and the Offshore Code Comparison Collaboration Continuation (OC4) projects [35, 38, 39]. Most current simulation tools considered in these efforts utilize reduced order equations that incorporate coefficients (e.g., added mass and damping coefficients) that are previously determined from scaled experiments or potential flow simulations [11, 27, 29, 39]. Engineering models that use coefficient-based methods have a rich history of application in ship hydrodynamics and the analysis of floating offshore structures such as oil platforms, the latter of which has motivated their use for simulating floating wind turbine platforms [19].

There are several software packages (e.g., AQWA [1], WAMIT [27], FAST [20, 21], and OrcaFlex [29]) available for time- and frequency-domain analyses of floating offshore structures using simplified physics-based methods, which include combinations of strip theory and panel methods [1, 13, 20, 25, 27, 29]. Such techniques neglect the detailed hydrodynamics and incorporate the fluid effects on the structure via coefficients in the six degree-of-freedom (6-DOF) equations of motion that account for the pressure field due to the motion of the body (added mass coefficient), damping effects (damping coefficient), buoyancy (restoring

coefficient) and wave excitation force vector [13, 25, 45]. In general, the coefficients are determined from scaled experiments (e.g., [2, 3, 45]) or higher-fidelity simulations (e.g., [6, 11, 39]), and are ideally restricted to the same geometry and conditions. Consequently, coefficient-based simulations can be inaccurate when extrapolated beyond the conditions used to determine the coefficients and under extreme conditions, such as the large amplitude, non-linear response of a floating offshore structure [15, 17, 38, 41].

Engineering models have been successfully applied in the analysis of offshore oil platforms [15, 17, 28, 36, 41] and offshore wind turbine platforms [7, 19, 38, 39]. Engineering models are invaluable for performing the numerous simulations required in the design of offshore wind turbines. However, offshore wind turbine systems must be designed to endure extreme conditions, where engineering models, particularly the underlying coefficients derived from scaled experiments or potential flow theory, may be less reliable [14, 38, 39]. For these design conditions, engineering models could benefit from the use of higher-fidelity modeling to provide more accurate coefficients.

High-fidelity methods such as computational fluid dynamics (CFD) combined with six-degree-of-freedom motion (6-DOF) simulations have some advantages over the use of engineering models. In particular, CFD/6-DOF simulations are capable of producing an accurate solution that requires only the geometry and mass properties of the platform, albeit at increased computational cost compared with engineering models [14]. A number of CFD/6-DOF solvers have been developed over the past 20 years and applied to store separation [32]; spacecraft atmosphere reentry [32]; aircraft, ship, and submarine motions [5, 40, 46]; and, more recently, offshore oil platform dynamics [22, 42, 47]. However, each of these solvers utilize closed-source or proprietary software to solve the Navier-Stokes equations and the 6-DOF equations of motion in a coupled fashion.

The objective of this study is to develop and validate a CFD/6-DOF solver using open-source software that can be used to accurately simulate floating offshore wind turbine platform dynamics. The open-source computational continuum mechanics library OpenFOAM [10] is used to develop a tightly coupled CFD/6-DOF solver, which is validated against benchmark experimental and theoretical data [16, 30]. The CFD/6-DOF solver is then applied to a semisubmersible floating offshore wind turbine platform from the DeepCwind [23, 24] and OC4 Phase II projects [39], and the results are compared with a FAST/HydroDyn engineering model simulation.

2. Material and Methods

Fluid structure interaction (FSI) solvers utilize either monolithic or partitioned algorithms. The monolithic approach simultaneously solves a single set of equations for both the solid and the fluid domains. The inherent stability and accuracy associated with the monolithic approach comes at the cost of code complexity, extreme memory requirements, and, in some cases, an ill-conditioned system matrix that is challenging to solve numerically [8]. Partitioned FSI solvers utilize separate solvers for the solid and fluid domains and are, thus, significantly simpler to develop and maintain [8, 9], as established and verified solid and fluid mechanics software can be combined using either a loosely or tightly coupled approach. This feature makes the partitioned FSI approach a popular choice [9].

Coupled CFD/6-DOF is a form of FSI, where the structure is non-deformable, and the motion of the solid is obtained by solving the Navier-Stokes and the 6-DOF equations of motion for the fluid and solid, respectively, in a coupled fashion using either a monolithic or partitioned algorithm. Such an approach is required to simulate the dynamics of floating offshore wind turbines, and to our knowledge a robust, open-source CFD/6-DOF solver does not exist for such free surface multiphase applications. The only such capability that we are aware of at this time is the standard multiphase CFD/6-DOF solver in OpenFOAM, interDyMFoam [10], which uses a loosely coupled partitioned approach that is often unstable. The instability is due to the loose coupling of the Navier-Stokes and 6-DOF equations of motion (see Algorithm 1), which introduces an artificial added mass that is independent of the discretization scheme [9, 12]. The term artificial in this case refers to non-physical added mass that is an artifact of the loose coupling between the CFD and 6-DOF solvers.

Algorithm 1 Loosely Coupled CFD/6-DOF Algorithm (interDyMFoam) [10]

1. Initialize variables
 2. Increment time
 - (a) Solve for body forces from fluid field
 - (b) Solve 6-DOF equations of motion for body
 - (c) Update the mesh
 - (d) Correct flux field variables to account for mesh motion
 - (e) Solve volume of fluid equation
 - (f) Solve for fluid pressure and velocity
-

Numerical instability due to artificial added mass is predominately affected by two parameters: density ratio and time step size [12]. If the density ratio ($\frac{\rho_f}{\rho_s}$) is sufficiently small (e.g., a heavy block falling through air), the effects of artificial added mass are insignificant, as the motion of the body is not appreciably affected by the fluid. For a body floating in water (e.g., offshore wind turbine platform), the density ratio is approximately unity, which results in significant artificial added mass effects. To illustrate the influence of time step size on the instability due to artificial added mass, we consider the classical benchmark problem of a heaving two-dimensional cylinder floating on a free surface [16, 30]. Here, the cylinder is initially displaced from its equilibrium position by $\frac{D}{3}$ in the heave direction (z), where it is released from at time $t = 0$ and the motion is allowed to decay over time (Figure 1).

Figure 1: Schematic illustration of free heave decay of a two-dimensional circular cylinder displaced by $D/3$ from its equilibrium position (blue region is water, red is air).

The benchmark simulation illustrates the effect of the time step size on the numerical stability of interDyMFoam, the standard loosely coupled multiphase CFD/6-DOF solver in OpenFOAM. Numerically, the solution stability is a complex interplay between the stability of the CFD solver, the 6-DOF solver, and the artificial added mass that is a result of the loose coupling. If the time step is too large ($\Delta t = 0.01$ s in Figure 2), the simulation diverges due to the inability to resolve the motion of the body and the fluid with such a coarse time step size. Similarly, if the time step is too small ($\Delta t = 0.0001$ s in Figure 2), solution divergence occurs near the point of peak acceleration due to a numerical instability that develops as a result of the increased artificial added mass associated with the use of a small time step size, as discussed by Förster et al. [12]. Though a stable solution can be obtained by optimizing the time step size to both damp the artificial added mass instability and to resolve the motion of the body and the fluid ($\Delta t = 0.001$ s in Figure 2), the optimum value is unknown a priori. Further, even if the solution remains stable, the error incurred from the artificial added mass in loosely coupled CFD/6-DOF simulations causes inaccuracies in the solutions of some quantities. For example, as shown in Figure 3, for the simulation that remains stable ($\Delta t = 0.001$ s), though the body displacement appears to be unaffected by the artificial added mass due to the body's inertia, the velocity and force solutions during peak acceleration exhibit highly oscillatory behavior that is non-physical.

Figure 2: Free heave decay of a two-dimensional circular cylinder with varying constant time step sizes using the standard loosely coupled multiphase CFD/6-DOF solver in OpenFOAM, interDyMFoam. Dots represent solution divergence

Figure 3: Solutions for (a) body displacement, (b) velocity, and (c) force from the simulation of free heave decay of a two-dimensional circular cylinder with a time step size of $\Delta t=0.001$ s using the standard loosely coupled multiphase CFD/6-DOF solver in OpenFOAM, interDyMFoam.

The instabilities due to artificial added mass inherent in a loosely coupled partitioned solver are well documented [9, 12], and a common solution is to implement a tightly coupled algorithm [8, 33, 34]. For example, the solver developed by Piro et al. [34] for simulating wave slamming on ship hulls, observed similar instabilities, which were ameliorated with the addition of tight coupling. The solution to these instabilities was to include a subiteration loop and dynamic relaxation to tightly couple the CFD and 6-DOF solutions [34]. Specifically, by implementing a subiteration loop around the 6-DOF and CFD solvers, the algorithm is tightly coupled, meaning that converged solutions for the body motion and fluid field are obtained at each time step before the simulation advances to the next time step (Algorithm 2) [9, 31]. Relaxation is applied to the 6-DOF body force (Equation 1) and moment (Equation 2) to stabilize the tightly coupled algorithm. To accelerate coupled solution convergence, dynamic relaxation is applied via the Aitken method [8, 26]

(Equation 5).

Beginning with the standard loosely coupled multiphase CFD/6-DOF solver in OpenFOAM, interDyM-Foam, we developed a tightly coupled solver based on Algorithm 2. Specifically, we implemented a subiteration loop and convergence checking within the time loop to ensure convergence of the CFD and 6-DOF solutions at each time step. In addition, to maintain solution stability and accelerate coupled solution convergence, we utilized the Aitken method to dynamically relax the 6-DOF body force and moment. The resultant tightly coupled CFD/6-DOF solver was then rigorously validated via a comparison with benchmark experimental data for free heave decay of a two-dimensional circular cylinder, and subsequently applied to the DeepCwind semisubmersible offshore floating wind turbine platform at full scale and compared with the NREL FAST/HydroDyn engineering tool. A coupled solution convergence criteria of 0.001 for the normalized force and moment residuals (Equations 3 & 4) was used for all simulations.

Algorithm 2 Tightly Coupled CFD/6-DOF Algorithm

1. Initialize variables
 2. Increment time
 - (a) Subiteration Loop
 - i. Solve for body forces from fluid field
 - ii. Relax body force and moment (Equations 1 and 2)
 - iii. Solve 6-DOF equations of motion
 - iv. Calculate the displacement and rotation of body
 - v. Update the mesh to follow body motion
 - vi. Correct flux field variables to account for mesh motion
 - vii. Solve volume of fluid equation
 - viii. Solve for fluid pressure and velocity
 - ix. Check coupled solution convergence (Equations 3 and 4)
 - x. Calculate new relaxation factor (Equation 5)
-

$$F_i^* = \omega_i \times (F_i - F_{i-1}) + F_{i-1} \quad (1)$$

$$M_i^* = \omega_i \times (M_i - M_{i-1}) + M_{i-1} \quad (2)$$

$$r_i^F = \frac{F_i - F_{i-1}}{m \times g} \quad (3)$$

$$r_i^M = \frac{M_i - M_{i-1}}{m \times g} \quad (4)$$

$$\omega_i = \omega_{i-1} \frac{(r_{i-1}^F)^T (r_i^F - r_{i-1}^F)}{|r_i^F - r_{i-1}^F|^2} \quad (5)$$

3. Results and Discussion

3.1. Verification and Validation

To illustrate the improved stability of the tightly coupled CFD/6-DOF solver, we performed simulations for the benchmark case of free heave decay of a two-dimensional circular cylinder. The increased stability of the tightly coupled solver is seen in Figures 4 and 5. In contrast to the loosely coupled CFD/6-DOF solver, the tightly coupled solver is stable for all time step sizes (Figure 4). Further, the non-physical oscillations in velocity and force are no longer present in the tightly coupled CFD/6-DOF solution (Figure 5).

Figure 4: Free heave decay of a two-dimensional circular cylinder with varying constant time step sizes using a custom tightly coupled multiphase CFD/6-DOF solver developed in OpenFOAM.

Figure 5: Solutions for (a) body displacement, (b) velocity, and (c) force from the simulation of free heave decay of a two-dimensional circular cylinder with a time step size of $\Delta t=0.001$ s using a custom tightly coupled multiphase CFD/6-DOF solver developed in OpenFOAM.

To verify the numerical simulations, we performed grid and time step convergence studies to ensure that the tightly coupled CFD/6-DOF solutions for free heave decay of a two-dimensional circular cylinder are independent of the mesh resolution and time step size. Specifically, we uniformly refined the computational mesh of the two-dimensional cylinder and performed simulations using three meshes: coarse, medium, and fine that contain approximately 50 thousand, 80 thousand, and 150 thousand computational cells, respectively. Simulations of free heave decay were performed and the solutions compared for each mesh. As shown in Figures 6 and 7, the tightly coupled CFD/6-DOF solutions are monotonically convergent with increasing mesh resolution (Figure 6) and decreasing time step size (Figure 7), with relatively small differences between each of the solutions. Quantitatively, to estimate the numerical error in the simulation results, we calculated the grid and time step convergence indices using the method described by Roache [37]. As shown in Table

1, the convergence indices for both amplitude and period are less than about 2.5%, thereby demonstrating that the numerical error in the computed results is small and the solution is insensitive to mesh resolution and time step size for the fine mesh and small time step case.

Figure 6: Grid convergence of normalized heave displacement for the benchmark case of free heave decay of a two-dimensional circular cylinder using a timestep of 0.0005 seconds

Figure 7: Time step convergence of normalized heave displacement for the benchmark case of free heave decay of a two-dimensional circular cylinder using the fine mesh

Table 1: Grid and time step convergence indices for the fine mesh and small time step case comparing the second peak (approximately 1.25 seconds)

	Period (%)	Amplitude (%)
Grid	1.33	0.0452
Time step	0.051	0.50

To validate our tightly coupled CFD/6-DOF solver, we performed simulations of the benchmark heaving cylinder case using the fine mesh and small time step and compared the results with theoretical solutions [30] and experimental data [16]. The theoretical solutions were derived by Ursell using potential theory [43, 44] to solve for the position of the freely heaving cylinder over time, and Ito [16] conducted an experiment that was designed to validate the theoretical solutions of Maskell and Ursell [30]. As shown in Figure 8, the present tightly coupled CFD/6-DOF solution closely matches the theoretical and experimental data. Quantitatively, the percent difference between the CFD/6-DOF solution and experimental data is small (less than about 3%; see Table 2), thereby validating our tightly coupled CFD/6-DOF solver.

Figure 8: Normalized free heave decay of a two dimensional circular cylinder from tightly coupled CFD/6-DOF, theory, and experimental data

Table 2: Percent difference between tightly coupled CFD/6-DOF solution and experimental data of Ito[16]

Peak	Period (%)	Amplitude (%)
1	1.46	1.52
2	0.822	3.03

3.2. Application to Semisubmersible

We then applied the validated tightly coupled CFD/6-DOF solver to the DeepCwind semisubmersible offshore floating wind turbine platform at full scale (Figure 9) and compared the results with the NREL FAST/HydroDyn engineering tool [20, 23, 24, 39]. To demonstrate both translation and rotational motion, free heave and pitch decay were simulated (see Figure 9). For the FAST simulations, we began with Test 25 of the FAST Certification Test cases and modified the input files to match the parameters and conditions used in the CFD/6-DOF simulations. This included turning off the mooring line and the filled members functionality. Ballast mass and mass moment of inertia were incorporated in the respective values for the platform. Additionally, as recommended for floating systems in the HydroDyn User’s Guide and Theory Manual [20], we calculated updated values for AddCLin using the equation provided in the manual. Four simulations in total were completed using both our tightly coupled CFD/6-DOF solver in OpenFOAM and FAST/HydroDyn. The simulations included free heave decay with one and six meter initial displacements above the equilibrium position in the vertical direction, and free pitch decay with four and eight degree initial rotation angles about the pitch axis of the platform, which is through its center of mass (Figure 9). For the free pitch decay simulations, the platform was pinned at its center of mass to isolate pitching motion

Figure 9: Schematic illustration of the DeepCwind semisubmersible offshore floating wind turbine platform at full scale as simulated in OpenFOAM

Prior to comparing with FAST, to ensure that the CFD/6-DOF simulations are accurate, we performed a grid and time step convergence study for the platform undergoing free heave decay with an initial displacement of 1 m. Specifically, we compared the CFD/6-DOF results for three meshes (coarse: 1.7 million cells, medium: 2.8 million cells, fine: 5.5 million cells) and three time step sizes (large: 0.01 seconds, medium: 0.005 seconds, small: 0.0025 seconds). As shown in Figures 10 and 11, the simulation results are insensitive to both mesh resolution and time step size at these levels of spatial and temporal discretization. This was confirmed quantitatively by computing the grid and time step convergence indices, which revealed that the magnitude of numerical error in the computed results for the fine mesh using the small time step size is extremely low (less than 1%; see Table 3)

Table 3: Grid and time step convergence indices for period and amplitude for the fine mesh and small time step size from CFD/6-DOF simulation results of free heave decay of the DeepCwind semisubmersible platform with a one meter initial displacement. The convergence indices were calculated from the CFD/6-DOF solutions at the start of the third period (at approximately 35 seconds).

	Period (%)	Amplitude (%)
Grid	0.0450	0.0124
Time step	0.178	0.190

Figure 10: Grid convergence of normalized heave displacement for the DeepCwind semisubmersible platform

Figure 11: Time step convergence of normalized heave displacement for the DeepCwind semisubmersible platform

The one meter heave decay case is initialized by displacing the platform one meter above equilibrium in the z direction and releasing it at time $t=0$. As shown in Figure 12, the tightly coupled CFD/6-DOF solution compares fairly well with the FAST simulation. Quantitatively, comparing the CFD/6-DOF and FAST solutions at the start of the second period (at approximately 17 s), the percent differences for the period and amplitude are approximately 2% and 10%, respectively. The comparison at other periods is comparable.

Figure 12: Comparison of heave displacement versus time between tightly coupled CFD/6-DOF and FAST/HydroDyn for free heave decay of the DeepCwind semisubmersible platform with a one meter initial displacement

The six meter heave decay case is initialized in a similar manner to the one meter heave case, but with the platform displaced by six meters above equilibrium. As shown in Figure 13, the overall comparison between tightly coupled CFD/6-DOF and FAST is relatively good. The largest discrepancy between the two solutions is in the amplitude of the first few periods of the heave decay, when viscous effects (e.g., flow separation from the offset columns, main column, pontoons, and cross braces) are expected to be most significant for such large amplitude motion. After the first few periods, when the platform displacements are smaller in amplitude, the comparison between CFD/6-DOF and FAST is much better. Quantitatively, the period comparison between the CFD/6-DOF and FAST solutions is quite close (less than about 1-2%). The percent difference in peak amplitude is about 38% at the maximum downward displacement (at approximately 9 seconds), and the difference in amplitude between the two solutions decreases at later times.

Figure 13: Comparison of heave displacement versus time between tightly coupled CFD/6-DOF and FAST/HydroDyn for free heave decay of the DeepCwind semisubmersible platform with a six meter initial displacement

For the four degree pitch decay simulation, the platform was initially rotated by four degrees about the pitch axis, which passes through its center of mass (see Figure 9), and released at time $t=0$ with all translational degrees of freedom fixed. As shown in Figure 14, the difference in the amplitudes between the CFD/6-DOF and FAST solutions is comparable to the one meter heave case, but the difference in the periods (about 4-5% difference) is slightly larger for the pitch decay than for the one meter heave decay. Given that the physics of pitching motion are more complex than heave due to the complex geometry of the platform, the present comparison between CFD/6-DOF and FAST is fairly good.

Figure 14: Comparison of pitch angle versus time between tightly coupled CFD/6-DOF and FAST/HydroDyn for free pitch decay of the DeepCwind semisubmersible platform with a four degree initial rotation angle

Simulation results for the eight degree pitch decay case are shown in Figure 15. In general, the comparison between the tightly coupled CFD/6-DOF and FAST results for the period is similar to the four degree pitch case, with percent differences of about 4-5% depending on the period. The difference in the amplitudes for the eight degree pitch case is slightly larger than for the four degree pitch decay case (percent differences of about 15-25% versus 5-15%, respectively), which is perhaps due to increased viscous or "shadowing" effects (e.g., see [4]) for the larger amplitude pitching motion. Even so, the comparison is reasonable.

Figure 15: Comparison of pitch angle versus time between tightly coupled CFD/6-DOF and FAST/HydroDyn for free pitch decay of the DeepCwind semisubmersible platform with an eight degree initial rotation angle

4. Conclusion

A tightly coupled CFD/6-DOF solver was developed using the computational continuum mechanics library OpenFOAM and validated via a comparison with theory and experimental data for the benchmark case of free heave decay of a two-dimensional circular cylinder. Tight coupling of the CFD and 6-DOF solvers via a subiteration loop and dynamic relaxation to accelerate coupled solution convergence was shown to be critical in eliminating the artificial added mass instability. Furthermore, the tight coupling was shown to eliminate spurious non-physical oscillations in velocity and force in CFD/6-DOF simulations that were not tightly coupled, but remained stable.

The validated tightly coupled CFD/6-DOF solver was then applied to the DeepCwind semisubmersible offshore floating wind turbine platform at full scale in simulations of free heave and free pitch decay. The solutions were compared with results from the NREL FAST/HydroDyn engineering tool. Overall, the CFD/6-DOF and FAST solutions compared reasonably well, particularly for cases with small amplitude displacements. In the larger amplitude (six meter) heave decay case, we observed greater discrepancy between the CFD/6-DOF and FAST results in the first few periods of heave decay and less discrepancy at later times, indicating that the discrepancy might be associated with Reynolds-number-dependent viscous effects. Preliminary FAST/HydroDyn simulations confirmed that by reducing the drag coefficient an improved comparison is obtained in the first few periods of large amplitude heaving motion. Additionally, in preliminary FAST/HydroDyn simulations, we observed that the results were highly dependent on the values used for AddCLin (the linear restoring matrix). Thus, though we manually calculated the AddCLin coefficients from the equations provided in the HydroDyn User's Guide and Theory Manual [20], these values may be adjusted to tune HydroDyn to match damping to experimental results, such as free-decay tests (or, in our case CFD/6-DOF results), as recommended in [20]. Even so, the present CFD/6-DOF and FAST/HydroDyn simulations, which did not include mooring lines, a tower, or wind turbine, compared reasonably well. In future

simulations that include these components, we anticipate that the differences between the CFD/6-DOF and FAST solutions observed in this study will be less significant. Indeed, preliminary simulations performed with FAST that included these components showed that they significantly affect the period and amplitude in free heave and pitch cases, to a much greater extent than the differences observed in the present study between CFD/6-DOF and FAST.

In future work, we plan to extend the present tightly coupled CFD/6-DOF solver to include mooring lines, waves, and wind turbine aerodynamics via an actuator line method [18], and continue validating the solver for increasingly complex configurations and operating conditions (e.g., using the experimental data acquired at the Maritime Research Institute Netherlands [23, 24]). Such a high-fidelity modeling capability may be used for wind turbine research, design, and analysis, and to complement and improve coefficient-based engineering tools (e.g., FAST) used to design offshore wind turbine platforms.

5. Acknowledgments

We thank Michael Lawson and Fabian Vorpahl for advice and recommendations in using FAST/HydroDyn. We also thank two anonymous reviewers for their insightful comments and suggestions on the manuscript. This work was carried out under the Penn State "Cyber Wind Facility" program DE-EE0005481 funded by the Department of Energy (DOE) under the U.S. Offshore Wind Technology Development program of the Office of Energy Efficiency and Renewable Energy (EERE). The research was partially funded by the Penn State Applied Research Laboratory under a Walker Assistantship.

6. References

- [1] ANSYS, Inc, 275 Technology Drive Canonsburg, PA 15317. *AQWA Reference Manual*, 14.5 edition, October 2012.
- [2] P Atsavapranee, J Carneal, D Grant, and A Percival. Experimental investigation of viscous roll damping on the DTMB model 5617 hull form. In *ASME 2007 26th International Conference on Offshore Mechanics and Arctic Engineering*, pages 561–570. American Society of Mechanical Engineers, 2007.
- [3] C Bassler, Carneal J, and Atsavapranee P. Experimental investigation of hydrodynamic coefficients of a wave-piercing tumblehome hull form. In *ASME 2007 26th International Conference on Offshore Mechanics and Arctic Engineering*, pages 537–548. American Society of Mechanical Engineers, 2007.
- [4] M Benitz, D Schmidt, M Lackner, G Stewart, J Jonkman, and A Robertson. Comparison of hydrodynamic load predictions between reduced order engineering models and computational fluid dynamics for the OC4-DeepCwind semi-submersible. In *ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering*, pages V09BT09A006–V09BT09A006. American Society of Mechanical Engineers, 2014.
- [5] M Bettle, A Gerber, and G Watt. Unsteady analysis of the six DOF motion of a buoyantly rising submarine. *Computers & Fluids*, 38(9):1833–1849, 2009.
- [6] L Bonfiglio, S Brizzolara, and C Chrysostomidis. Added mass and damping of oscillating bodies: a fully viscous numerical approach. *Recent Advances in Fluid Mechanics, Heat & Mass Transfer and Biology*, ISBN, pages 978–1, 2011.
- [7] C Butterfield, W Musial, J Jonkman, P Sclavounos, and L Wayman. Engineering challenges for floating offshore wind turbines. 2007.
- [8] R Campbell. *Fluid-Structure Interaction and Inverse Design Simulations for Flexible Turbomachinery*. PhD thesis, The Pennsylvania State University, 2010.
- [9] P Causin, J Gerbeau, and F Nobile. Added-mass effect in the design of partitioned algorithms for fluid-structure problems. *Computer methods in applied mechanics and engineering*, 194(42):4506–4527, 2005.

- [10] Open CFD. OpenFOAM user guide. *OpenFOAM Foundation*, 2.2.x, 2011.
- [11] G Contento. Numerical wave tank computations of nonlinear motions of two-dimensional arbitrarily shaped free floating bodies. *Ocean engineering*, 27(5):531–556, 2000.
- [12] C Förster, W Wall, and E Ramm. Artificial added mass instabilities in sequential staggered coupling of nonlinear structures and incompressible viscous flows. *Computer methods in applied mechanics and engineering*, 196(7):1278–1293, 2007.
- [13] T Fossen. *Handbook of marine craft hydrodynamics and motion control*. John Wiley & Sons, 2011.
- [14] A Henderson and M Patel. On the modelling of a floating offshore wind turbine. *Wind Energy*, 6(1):53–86, 2003.
- [15] N Hogben, G Ward, LH Carpenter, BL Morison, JR Miller, JW Searle, and GH Keulegan. Estimation of fluid loading on offshore structures. In *ICE Proceedings*, volume 63, pages 515–562. Ice Virtual Library, 1977.
- [16] S Ito. *Study of the transient heave oscillation of a floating cylinder*. PhD thesis, Massachusetts Institute of Technology, 1977.
- [17] M Jameel, S Ahmad, A Islam, and M Jumaat. Non-linear dynamic analysis of coupled spar platform. *Journal of Civil Engineering and Management*, 19(4):476–491, 2013.
- [18] P Jha, M Churchfield, PJ Moriarty, and S Schmitz. Accuracy of state-of-the-art actuator-line modeling for wind turbine wakes. *AIAA Paper*, (2013-0608), 2013.
- [19] J Jonkman. *Dynamics modeling and loads analysis of an offshore floating wind turbine*. ProQuest, 2007.
- [20] J Jonkman. HydroDyn manual. *NWTC Information Portal (HydroDyn)*. <https://nwtc.nrel.gov/HydroDyn>. Last modified 25-March-2015 ; Accessed 24-July-2015, 2013.
- [21] J Jonkman and L Buhl, Jr Marshall. FAST users guide. *NWTC Information Portal (FAST v8)*. <https://nwtc.nrel.gov/FAST8>. Last modified 22-September-2015 ; Accessed 29-September-2015, 2005.
- [22] J Kim, A Magee, and K Guan. CFD simulation of flow-induced motions of a multi-column floating platform. In *ASME 2011 30th International Conference on Ocean, Offshore and Arctic Engineering*, pages 319–326. American Society of Mechanical Engineers, 2011.
- [23] RW Kimball, AJ Goupee, AJ Coulling, and HJ Dagher. Model test comparisons of TLP, spar-buoy and semi-submersible floating offshore wind turbine systems. In *Proc. 2012 SNAME Annual Meeting and Expo*, 2012.
- [24] B Koo, A Goupee, K Lambrakos, and R Kimball. Model tests for a floating windturbine on three different floaters. In *ASME 2012 31st International Conference on Ocean, Offshore and Arctic Engineering*, pages 455–465. American Society of Mechanical Engineers, 2012.
- [25] B Korvin-Kroukovsky. *Theory of seakeeping*. Society of Naval Architects and Marine Engineers, 1961.
- [26] U Küttler and W Wall. Fixed-point fluid–structure interaction solvers with dynamic relaxation. *Computational Mechanics*, 43(1):61–72, 2008.
- [27] CH Lee and JN Newman. WAMIT® user manual, versions 6.3, 6.3 pc, 6.3 s, 6.3 s-pc. *WAMIT, Inc., Chestnut Hill, MA, USA*.
- [28] YM Low and RS Langley. A hybrid time/frequency domain approach for efficient coupled analysis of vessel/mooring/riser dynamics. *Ocean Engineering*, 35(5):433–446, 2008.
- [29] Orcaflex Manual. Online at <http://www.orcina.com/softwareproducts/orcaflex/documentation>. *OrcaFlex.pdf*.

- [30] SJ Maskell and F Ursell. The transient motion of a floating body. *J. Fluid Mech*, 44(part 2):303–313, 1970.
- [31] H Matthies and Steindorf J. Partitioned but strongly coupled iteration schemes for nonlinear fluid–structure interaction. *Computers & structures*, 80(27):1991–1999, 2002.
- [32] SM Murman and MJ Aftosmis. Dynamic analysis of atmospheric-entry probes and capsules. *AIAA Paper*, 74, 2007.
- [33] E Paterson, R Wilson, and F Stern. General-purpose parallel unsteady RANS ship hydrodynamics code: CFDSHIP-Iowa. Technical report, DTIC Document, 2003.
- [34] D Piro and K Maki. Hydroelastic analysis of bodies that enter and exit water. *Journal of Fluids and Structures*, 2012.
- [35] W Popko, F Vorpahl, A Zuga, M Kohlmeier, J Jonkman, A Robertson, T Larsen, A Yde, K Sætertrø, K Okstad, et al. Offshore code comparison collaboration continuation (OC4), phase I—results of coupled simulations of an offshore wind turbine with jacket support structure. In *22nd International Society of Offshore and Polar Engineers Conference. Rhodes, Greece*, 2012.
- [36] Z Ran, MH Kim, JM Niedzwecki, RP Johnson, et al. Responses of a spar platform in random waves and currents (experiment vs. theory). *International Journal of Offshore and Polar Engineering*, 6(01), 1996.
- [37] Patrick J Roache. *Verification and validation in computational science and engineering*. Hermosa, 1998.
- [38] A Robertson, J Jonkman, A Goupee, A Coulling, I Prowell, J Browning, M Masciola, and P Molta. Summary of conclusions and recommendations drawn from the DeepCwind scaled floating offshore wind system test campaign. In *ASME 2013 32nd International Conference on Ocean, Offshore and Arctic Engineering*, pages V008T09A053–V008T09A053. American Society of Mechanical Engineers, 2013.
- [39] A Robertson, J Jonkman, M Masciola, H Song, A Goupee, A Coulling, and C Luan. Definition of the semisubmersible floating system for phase ii of OC4. *NREL report*, 2012.
- [40] T Schwarz. RANS simulations with one and six degrees of freedom rigid body motions. In C. Tropea et al., editor, *New Res. in Num. and Exp. Fluid Mech. VI*, pages 178–185, Berlin, Heidelberg, 2007. Springer-Verlag.
- [41] MR Tabeshpour, AA Golafshani, and MS Seif. The effect of added mass fluctuation on heave vibration of TLP. *International Journal of Engineering-Materials and Energy Research Center-*, 18(3):219, 2005.
- [42] J Tan, A Magee, J Kim, Y Teng, and N Zukni. CFD simulation for vortex induced motions of a multi-column floating platform. In *ASME 2013 32nd International Conference on Ocean, Offshore and Arctic Engineering*, pages V007T08A066–V007T08A066. American Society of Mechanical Engineers, 2013.
- [43] F Ursell. On the heaving motion of a circular cylinder on the surface of a fluid. *The Quarterly Journal of Mechanics and Applied Mathematics*, 2(2):218–231, 1949.
- [44] F Ursell. On the rolling motion of cylinders in the surface of a fluid. *The Quarterly Journal of Mechanics and Applied Mathematics*, 2(3):335–353, 1949.
- [45] JH Vughts. The hydrodynamic coefficients for swaying, heaving and rolling cylinders in a free surface. Technical report, 1968.
- [46] R Wilson, P Carrica, and F Stern. Unsteady RANS method for ship motions with application to roll for a surface combatant. *Computers and Fluids*, 35(5):501 – 524, 2006.
- [47] Q Xu, J Kim, T Bhaumik, J OSullivan, and J Ermon. Validation of HVS semisubmersible VIM performance by model test and CFD. In *ASME 2012 31st International Conference on Ocean, Offshore and Arctic Engineering*, pages 175–185. American Society of Mechanical Engineers, 2012.