J. Wind Eng. Ind. Aerodyn. 147 (2015) 104–119

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia

The coupled dynamic response computation for a semi-submersible platform of floating offshore wind turbine

Thanh Toan Tran, Dong-Hyun Kim*

Graduate School of Mechanical and Aerospace Engineering, Research Center for Offshore Wind Turbine Technology (ReCOWT), Gyeongsang National University (GNU), 900 Gajwa-dong, Jinju 660-701, South Korea

ARTICLE INFO

Article history:
Received 18 July 2014
Received in revised form
25 September 2015
Accepted 26 September 2015

Keywords:
Floating offshore wind turbine (FOWT)
Semi-submersible platform
Overset grid approach
Morison equation
Free-decay analysis
OC4 DeepCWind

ABSTRACT

Many numerical methods based on potential-flow theory have been used to analyze the hydrodynamic effect of floating offshore wind turbine (FOWT). However, they cannot directly and fully consider the complex viscous interference effects for the motion of multi-structure platform geometry. To accurately account for this effect, it necessarily requires an additional numerical modification process based on experimental data or an implementation of Morison equation. This paper deals with an unsteady hydrodynamic simulation using CFD method with dynamic motion based on overset grid and potential based panel approach for the semi-submersible DeepCWind FOWT. Using the CFD approach, the coupled fluid flow and multi-body dynamic analysis is applied using the volume of fluid approach to investigate the hydrodynamic responses of the typical semi-submersible floater. Herein, the restoring force and moment of the catenary mooring lines are also considered in the time-domain. Additionally, a potential flow linear diffraction model extended with and without Morison elements to include viscous damping is performed. It is shown that the present results show good correlations with experimental data without using the adjusting parameters. In addition, numerical tests for major solver parameters of the CFD method are conducted, verified, and investigated in detail.

1. Introduction

The offshore wind turbines are being more attracted to researchers, engineer as well as the universities, institutes, or governments. Various Multi-MW turbine systems have been installed in the offshore area since the year 2000 (EWEA, 2015). Until the end of June 2015, total amount of installed offshore wind turbine (OWT) rises up to 3072 units, with a combined capacity of 10,393.6 MW fully grid connected in European waters in 82 wind farms across 11 countries (EWEA, 2015). One of the reasons may be explained that a wind speed is typically stronger and much more sustained in the offshore wind farm (OWF). The developments of the offshore wind turbine associated with the fixed support platforms, which are based on the experiences of the onshore wind turbines, have led to the installation of the OWF in the shallow water. On the other hand, the floating offshore wind turbines (FOWT) can be installed even in the deep sea area based on the existing technology, the construction experience of offshore petroleum and natural gas industries for the design and installation of the supporting platform. In the viewpoint of engineering design, the FOWT have several difficulties such as more advanced blade control due to the floating motion; the large inertia loading on the tower and nacelle caused by the induced accelerations due to floater motions; and more expensive and complicated installation processes, etc. (Luo et al., 2012 and Transportation Research Board, 2011). If above issues can be efficiently solved, the FOWT farms are expected to generate a large amount of clean energy with a competitive price compare to other energy resources. Thus, the design, manufacturing, installation, control, and maintenance of the FOWT still have many challenges.

One of the common challenges to all structure designs of FOWT is the ability to predict the dynamic load responses of the coupled wind turbine and platform system which usually combines a wind loading and a stochastic wave. However, the combined dynamic characteristics of the load components are still not well understood and very hard to accurately calculate because of their complex multi-physical phenomena under realistic operating conditions. Theory and numerical analysis methods are necessarily developed and validated to optimize the FOWT design. Because of the load prediction challenges for design requirements, various experimental floating substructures have been performed. At the present, several experimental floating substructures (for example, four in Europe, two in Japan and one in the US) are completely finished or to be in test phases: SeaTwirl, SWAY, Blue H and Poseidon in Europe, Kabashima Island concept and WindLens in Japan and DeepCwind floating turbine in the US (EWEA, 2015 and Maine

* Corresponding author.
E-mail addresses: dhk@gnu.ac.kr, dhk0521@gmail.com (D.-H. Kim).

http://dx.doi.org/10.1016/j.jweia.2015.09.016
0167-6105/

International Consulting, 2013). In particular, the scale model tests for a floating offshore wind turbine have been performed (Coulling et al., 2013 and Goupee et al., 2012). However, an experiment of the floating offshore wind turbine usually is more expensive than a sophisticated design tool which is preferred to cost effective solutions of an FOWT during a design process. Therefore, the sophisticated modeling tools that simulate the whole behavior of the structure should be developed and validated to allow for an improved design (EWEA, 2013).

Theoretically, the effect of viscous flow in the hydrodynamic simulation of a floating offshore platform could not be directly included by the potential based panel approaches (Coulling et al., 2013; Jonkman, 2009, 2010; Masciola et al., 2013; Gueydon and Weller, 2013). It needs to be corrected using the model of Morison's formulation based on strip theory or an additional damping coefficient based on experimental test data (Coulling et al., 2013; Masciola et al., 2013, and Goupee et al., 2012). These approaches have been used by a well-known software code for floating wind turbine, NREL FAST which is computationally time efficient. Recently, the HydroDyn module of NREL FAST code has been extended to allow for multiple approaches for calculating the hydrodynamic loads on a structure: a potential-flow theory solution, a strip-theory solution, or a combination of the two (Jonkman et al., 2014). In particular, the relative form of Morison's equation for the distributed fluid-inertia, added-mass, and viscous-drag components has been included in the strip-theory loads. However, it inherently requires a third-party module such the diffraction/radiation panel program WAMIT (Lee et al., 1991) that provides the added mass matrix, the matrix of hydrostatic restoring force, and the matrix of retardation for the platform. It has been widely used as a pre-processor for the floater coefficient that is also employed by the floater-mooring coupled dynamic program such as the CHARM3D (Bae and Kim, 2013). However, this is commonly based on a panel method in the frequency-domain. Some physical phenomena such as wave run-up against the semi-submersible columns (Shan et al., 2011 and Diaconu, 2013) and viscous effects on the floater cannot be adequately captured yet by the conventional models. Additionally, the results of such simulations can only be trusted if these are validated by the scale model experiments (Huijs et al., 2014a). Whereas, the unsteady CFD approach can directly include all related physical effects (flow viscosity, hydrostatic, wave diffraction, radiation, wave run-up, and slamming, etc.) of the floating platform (Christensen et al., 2005; Ramirez et al., 2011, and Chen and Yu, 2009). Additionally, several CFD simulations have been performed to investigate the hydrodynamic characteristics of FOWT platform. The OC3 Hywind platform model, of which geometry is relatively simple compared to semi-submersible platform was considered and analyzed in the previous studies using CFD (Beyer et al. 2013; Quallen et al., 2013). However, the heave DOF motion was constrained to prevent numerical instabilities during the first iterations of the coupled simulation (Beyer et al. 2013). The lack of mooring line solver in naoe-FOAM-SJTU solver is being developed (Zhao and Wan, 2015); or the simulation of semi-submersible with free six-degrees of freedom (6-DOFs) body motion is further conducted by NREL to thoroughly compare to experimental results from the DeepCwind project (Benitz et al., 2014, 2015). The OPENFOAM code will be applied to supplement the experimental data set by simulating the extreme conditions, not simulated in the wave tank (Benitz et al., 2014). Further simulations for the complex geometries of an offshore wind turbine platform are necessarily performed to validate and investigate the dynamic characteristics due to the hydrodynamic loads.

The major purpose of the present study is to show the accurate prediction of hydrodynamic responses of a semi-submersible platform of the floating offshore wind turbine. The advanced computational fluid dynamic (CFD) approach with a dynamic overlapping grid technique is applied to account for the dynamic motion of the floating platform in viscous flows. In this study, the semi-submersible floating platform such as OC4 DeepCWind which consists of multiple columns and partially connected beams is considered. The fluid flow and multibody dynamics (MBD) coupling analysis method has been applied in conjunction with the volume of fluid (VOF) modeling in order to accurately investigate the hydrodynamic responses and unsteady viscous flows around the floating platform in detail. Herein, a quasi-static mooring line which can provide the induced hydrodynamic forces to the supporting platform in the time-domain is also considered. The effect of viscous interferences for an elaborate platform configuration has been carried out using the advanced turbulence model. Additionally, a potential flow theory with a linear diffraction model extended with and without Morison elements has been performed to investigate the effect of viscous damping and viscous wave excitation. The results of the present study are compared with experimental data which was already carried out at MARIN (Maritime Research Institute Netherlands) for a 1:50 Froude scaling model (Coulling et al., 2013). The tests were performed by the University of Maine DeepCWind Consortium. The comparison results for free-decay response analyses showed overall good correlations among the CFD, the AQWA, the FAST, the Simo/Riflex, and the experiment. Moreover, the CFD simulation results for regular wave conditions show very good agreements with the experimental data. It is shown that the advanced CFD method can provide accurate hydrodynamic responses without adjusting parameters that are commonly required in the conventional potential-based approaches. In addition, the unsteady analyses using the CFD method have been performed for the cases of inviscid, laminar, and turbulent flows. Furthermore, to show the effect of different turbulence models in the CFD analysis, various computations considering the Spalart–Allmaras, k–ε, and k–ω models are conducted and the results are investigated in detail.

2. Simulation method

The present study is based on an application and analysis of computational fluid dynamics with overset grid technique and conventional potential based panel method. The three-dimensional, unsteady Reynolds-averaged Navier–Stokes (RANS) with several turbulent models including Spalart–Allmaras (S–A), k–ω shear stress transport (SST), standard k–ε are performed based on the STAR-CCM+ software (Ver.9.06). For potential based panel analysis, three-dimensional radiation/diffraction theory has been performed by the commercial ANSYS/AQWA (Ver.14.0) package. The viscous effect has been investigated with and without using extended Morison element. The quasi-static and dynamic catenary mooring lines have been also considered, herein. Additionally, the latest version of FAST code (Version 8.10) with the use of the mooring analysis program (MAP) option has been applied to verify and compare the current simulation results. The second order wave loads have been used by the FAST simulation.

2.1. Computational fluid dynamic method

In the present study, the basic flow equations which are an integral form of Navier–Stokes equations have been applied. The segregated flow model is applied to solve the flow equations (one for each component of velocity, and one for pressure). All simulations are employed a semi-implicit method for pressure-linked equations (SIMPLE) solution algorithm which the linkage between the momentum and continuity equation is achieved with a predictor-corrector approach. The second-order upwind scheme is applied for convection term. This method has been shown to be an efficient means of solving the incompressible Reynolds-averaged Navier–Stokes (RANS) equations. In the unsteady simulations, both first and second-order central difference scheme is used for temporal time discretization. Additionally, the overset based technique has been applied to handle the complex geometries and the relative motion of bodies in dynamic