Ocean Engineering xxx (2017) 1–12

Contents lists available at ScienceDirect

Ocean Engineering

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

Validation and application of a fully nonlinear numerical wave tank for simulating floating offshore wind turbines

N. Bruinsma *, B.T. Paulsen, N.G. Jacobsen

Department of Hydraulic Engineering, Deltares, Boussinesqweg 1, 2629 HV Delft, The Netherlands

ARTICLE INFO

Keywords:
Floating offshore wind turbines
OpenFOAM
Coupled CFD/6-DOF
Waves2Foam
Mooring lines

ABSTRACT

Numerical models are becoming a valid supplement, and even a substitute, to physical model testing for the investigation of fluid-structure interaction due to improved methods and a continuous increase in computational power. This research presents an extensive validation of a fully nonlinear numerical wave tank for the simulation of complex fluid-structure interaction of moored floating structures. The validation is carried out against laboratory measurements including recent and unpublished laboratory measurements of the OC5 floating offshore wind turbine subjected to waves. The numerical wave tank is based on the Navier-Stokes/6-DOF solver, interDyMFoam, provided by the open-source CFD-toolbox OpenFOAM, extended with the wave generation and absorption toolbox, waves2Foam, and an implementation for restraints of floating structures. Two methods are evaluated to address the instability issues of the partitioned Navier-Stokes/6-DOF solver, which are associated with artificial added mass. The model has been shown capable of computing detailed fluid-structure interactions, including dynamic motion response of a rigid structure in waves. However, instabilities due to numerical added mass is observed and discussed and it is concluded that further research is needed in order to establish a efficient yet stable scheme. Overall, good agreement is achieved between the numerical model and the physical model results.

1. Introduction

In recent years a transformation in energy supply from burning of fossil fuels towards renewable energy sources such as solar, wind and hydropower is seen. In northern Europe this change is mainly driven by wind energy and lately also offshore wind energy. The increase in wind energy capacity is expected to continue for several decades partly by replacing existing capacity with newer, larger and more efficient turbines, but for the greater part by the installation of new wind farms (Pineda et al., 2014). Offshore wind energy installations are expected to produce approximately 2.9% of the total European Union energy demand by 2020. This foreseen growth will move offshore wind energy from an emerging and immature technology to a key component of the energy mix (Arapogianni et al., 2011).

Current developments of offshore wind are primarily based on bottom mounted foundations and the vast majority of those on monopile foundations. Gravity based substructures are the second most common foundation type, followed by jacket structures. However, many countries have limited suitable sites in sufficiently shallow water to economically install offshore wind turbines on bottom mounted substructures. Floating offshore wind turbine (FOWT) concepts are seen as a promising solution to unlock the offshore potential in deeper waters. This technology is still at an early stage of development and well validated modelling tools capable of simulating dynamic behaviour are necessary to improve the design of these floating structures (Arapogianni et al., 2013). One of the challenges for FOWT at intermediate water depth ($50\text{ m} < h < 120\text{ m}$) is optimizing the dynamic behaviour of the mooring system while retaining a limited footprint. Optimization of mooring systems for semi-submersibles and FOWT foundations are discussed in Robertson et al. (2011); Brommundt et al. (2012); Huijs (2015). The common approach for these studies is to use numerical models to analyse different load cases including the effect of wind and wave. However not discussed in this paper, the presented and validated numerical model is suitable for such analysis and one of the reasons why this research was initialized.

The International Energy Agency has facilitated a series of international collaboration projects to improve offshore wind modelling tools. These projects aim to verify and validate the accuracy of complex engineering tools through code-to-code and code-to-data comparisons (Jonkman and Musial, 2010; Popko et al., 2012; Robertson et al., 2014b). The most recent of these projects is the Offshore Code Comparison

* Corresponding author.
E-mail address: niek.bruinsma@deltares.nl (N. Bruinsma).

https://doi.org/10.1016/j.oceaneng.2017.09.054
Received 18 January 2017; Received in revised form 17 August 2017; Accepted 24 September 2017
Available online xxxx
0029-8018/

Collaboration, Continued, with Correlation (OC5). This project continues with the improvement of modelling tools based on the dynamic analysis of the DeepCwind design (Koo et al., 2014; Robertson et al., 2014a). This semi-submersible design features a 5 MW horizontal-axis wind turbine with pitch controlled blades developed by the National Renewable Energy Laboratory (Jonkman et al., 2009; Popko et al., 2012).

The simulation tools applied in these research projects are frequency-domain potential flow models such as; AQWA (ANSYS, 2010) and WAMIT (Lee and Newman, 2006). These lower order numerical models are used to determine the frequency-dependent added mass and damping coefficients. The output from these models is used in a time-domain model, e.g FAST (Jonkman et al., 2014), to perform the transient analysis. Especially for the analysis of a FOWT in extreme conditions, where these lower order models may be less reliable, it could be beneficial to use a higher order numerical model for describing the fluid-structure interaction.

Higher order numerical models, like Computational Fluid Dynamics (CFD) combined with 6-DOF simulations, have been successfully used as a tool in many areas of engineering. With CFD models it is possible to use higher order wave models and even accurately simulate wave breaking. The major benefit of using a higher order CFD/6-DOF model is the capability of solving the fluid-structure interaction problem based on the geometry and mass properties of the floating structure, without predetermining coefficients. However, the accuracy of predictions and numerical convergence needs to be validated in order to use CFD/6-DOF models as a reliable design tool for floating offshore structures.

The objective of this research is to establish a well validated fully nonlinear numerical wave tank for the accurate simulation of complex fluid-structure interaction of moored floating offshore structures using a higher order CFD/6-DOF model based on the opensource CFD-toolbox OpenFOAM (Weller et al., 1998). Section 2 provides an overview of the numerical models used to set up the numerical wave tank. In Section 3 the numerical model is validated against theoretical and experimental data. This validation consists of three approaches. First, a wave load and decay case with a two-dimensional cylinder are presented, where the model is validated against benchmark experimental and theoretical data from Dixon et al. (1979), Ito (1977) and Maskell and Ursell (1970). Secondly, the model is used to simulate three-dimensional moored decay of a FOWT from the OC5 projects. Lastly, the moored FOWT is subjected to regular incoming waves. Both the decay and wave loading cases with the OC5 FOWT are validated against physical experiments. These tests were carried out in the concept basin at MARIN and besides focusing on hydrodynamic loading also the effect of the wind turbine controller on the dynamics of the floater was investigated.

2. Material and methods

The multiphase interDyMFoam solver of OpenFOAM is a segregated fluid-structure interaction solver, where the flow-dependent motions of a rigid body are obtained by solving the Navier-Stokes and 6-DOF equations in a coupled manner. The coupled Navier-Stokes/6-DOF solver has an unstable nature, as discussed in Seng (2012) and Dunbar et al. (2015). In the present work, two different methods for stabilizing such a solver are evaluated. Here the standard interDyMFoam solver, provided by the opensource CFD-toolbox OpenFOAM (Weller et al., 1998), version 2.3.1., is extended with the wave generation and absorption toolbox waves2Foam, developed by Jacobsen et al. (2012). This combination is referred to as the waveDyMFoam solver. Furthermore, an implementation for the restraints of floating structures was developed. The structure of this section is as follows: first the governing equations for the Navier-Stokes/Volume-Of-Fluid (VOF) solver are presented in Section 2.1, the differences between the two stabilizing methods are described in Section 2.2, in Section 2.3 the restraints implementation is presented, the applied boundary conditions are presented in Section 2.4, wave generation and absorption with relaxation zones is described in Section 2.5 and finally the spatial and temporal discretization is described in Section 2.6.

2.1. Governing equations Navier-Stokes/VOF

The waveDyMFoam solver utilizes the two-phase incompressible Navier-Stokes equations in combination with a VOF-surface capturing scheme (Hirt and Nichols, 1981) to compute fluid structure interaction. The governing equations, used in the Navier-Stokes/VOF solver, for conservation of mass and momentum of an incompressible flow of air and water are given by

$$\nabla\cdot\mathbf{u} = 0, \tag{1}$$

$$\frac{\partial \rho\mathbf{u}}{\partial t} + \nabla\cdot(\rho\mathbf{u})\mathbf{u}^T = -\nabla p^* + (\mathbf{g}\cdot\mathbf{x})\nabla\rho + \nabla\cdot(\mu\nabla\mathbf{u}), \tag{2}$$

where $\nabla = (\partial_x, \partial_y, \partial_z)$ is the three-dimensional gradient operator, $\mathbf{u} = (u, v, w)$ is the velocity field in Cartesian coordinates, $\mathbf{g}$ is the gravitational acceleration and p^* is pressure in excess of the hydrostatic pressure, which relates to the total pressure, p, by

$$p^* = p - \rho(\mathbf{g}\cdot\mathbf{x}). \tag{3}$$

Furthermore, the local density, ρ, and viscosity, μ, are given by the water volume fraction, α, consistent with

$$\rho = \alpha\rho_{water} + \rho_{air}(1-\alpha), \tag{4}$$

$$\mu = \alpha\mu_{water} + \mu_{air}(1-\alpha), \tag{5}$$

where α is zero for air, one for water and a mixture of the two for all intermediate values.

OpenFOAM uses a VOF method for tracking the air-water interface. After obtaining the velocity field by solving equations (1) and (2) for the two-phase flow of air and water, the VOF method (Hirt and Nichols, 1981) can be used to advance the α field in time with the following scalar advection equation

$$\frac{\partial\alpha}{\partial t} + \nabla\cdot\mathbf{u}\alpha + \nabla\cdot\mathbf{u}_r\alpha(1-\alpha) = 0. \tag{6}$$

Using a standard finite-volume approximation for solving the hyperbolic advection equation (6) would lead to significant smearing of the interface. This is significantly reduced by the introduction of an interface compression term as discussed in Berberović et al. (2009). In equation (6) this is the last term of the left-hand side. The interface compression term is only active in the vicinity of the interface, $0 < \alpha < 1$, where its strength is governed by the relative velocity, u_r. It is stressed that even though u_r has the dimension of m/s, it lacks any physical meaning. To ensure stability, a multi-dimensional flux limited scheme (MULES) is used for solving the scalar advection equation (6).

2.2. Stability of a coupled Navier-Stokes/6-DOF solver

In order to simulate the flow-dependent motion response of a floating structure, a partitioned fluid-structure interaction model is used. Here, the rigid body motion is obtained by separately solving the Navier-Stokes and 6-DOF motion equations for the fluid and the solid.

For numerical models with a loos coupling between the fluid solver and the motion of the body, the discrete time integration may become unstable leading to non-physical motion of the object. Effectively these instabilities appear as a numerical added-mass and hence change the behaviour of the structure. If the density difference between the fluid and the body becomes smaller, the fluid has a larger influence on the motions of the body and numerical simulations are more prone to instabilities. The temporal discretization of the numerical simulation was also found to have a significant effect on the stability of the solution. Thorough discussions on instabilities in partitioned fluid-structure interaction models as a result of artificial added mass can be found in the work by