Hydrodynamic validation of a semi-submersible floating platform supporting a 15MW wind turbine tower under extreme loading scenarios with DualSPHysics and MoorDyn+

B. Tagliaferro,1 S. Capasso,2 I. Martinez-Estèvez,3 M. Göteman,4 H. Bernhoff,4 M. Karimirad,5 JM. Domínguez,3 C. Altomare,1 G. Viccione,2 AJC. Crespo,3 M. Gómez-Gesteira3

1Laboratori d'Enginyeria Marítima, Universitat Politècnica de Catalunya - BarcelonaTech (UPC) Barcelona, Spain
2Department of Civil Engineering, University of Salerno, Italy
3Environmental Physics Laboratory, CIM-UVIGO, Universidade de Vigo, Spain
4Dept. of Electrical Engineering, Uppsala University, Sweden
5School of Natural and Built Environment, Queen’s University Belfast, UK

ABSTRACT

To investigate the hydrodynamic performance of the floating platform VolturnUS-S as configured for the 1st Floating Offshore Wind Turbine (FOWT) Comparative Study, we have used a Smoothed Particle Hydrodynamics (SPH) based solver that features a coupling to the cable dynamic solver MoorDyn+ to reproduce the proposed benchmarks. This is a quite novel application of the method to simulate semi-submersible platforms for offshore wind energy. For this benchmark, which does not include aerodynamic actions, we have proposed a new procedure, leveraging offline coupling techniques to model the problem in a sub-domain of the reference wave basin. Our approach is detailed and validated for wave propagation only, and thus applied to reproduce the wave-platform interaction for an extreme focused wave condition. Good results are obtained for the wave generation and validation using open boundary conditions as well as for the platform motion under the extreme event.

KEY WORDS: CFD; MESHIN; DualSPHysics; SPH; Focused wave; FOWT.

INTRODUCTION

The 1st Floating Offshore Wind Turbine (FOWT) Comparative Study proposes a framework for evaluating strengths and weaknesses of numerical approaches in dealing with the hydrostatic and hydrodynamic response of semi-submersible platforms for wind energy harvesting. For this investigation, the VolturnUS-S floating platform (Allen et al., 2020), which ideally hosts a IEA-15-240-RWT wind turbine (Gaertner et al., 2020) serves as a reference test configuration. Several test cases are available in the reference material (Ransley et al., 2023). The participants are required to submit the output data for the model they want to benchmark.

Developed by researchers from the National Renewable Energy Laboratory (NREL), FAST (Fatigue, Aerodynamics, Structures and Turbulence) (Jonkman, 2007) is certainly one of the most used open-source codes for marine structures. FAST and similar pieces of software, however, solve the hydrodynamics with potential-flow solvers, which generally cannot properly capture viscous effects and wave breaking. Anyhow, most of their limitations can be worked around by including fictitious treatments for the viscosity and other effects. Tran et al. (2014) compared the prediction of FAST and a computational fluid dynamics (CFD)-based solver, and revealed that potential-flow based models consistently overestimate the motion of the investigated platforms (see also, Nemathaksh et al., 2015; Ogun et al., 2018; Zhou et al., 2019). Similar comprehensive research is presented in Liu et al. (2017), where another CFD investigation was performed on the DeepCWind semi-submersible floating wind turbine within a realistic-like environment. Other researchers have proposed the use of combined design strategies (see, e.g., for similar applications Rij et al., 2019; Ma et al., 2019) in which CFD models can play a pivotal role.

Among the CFD methods, Meshfree Particle Methods (MPMs) stand out for several advantages in simulating free-surface flows and their interaction with fixed or moving structures over mesh-based methods (see, e.g., Rakhsha et al., 2021; Wang et al., 2022), as the computational nodes move according to the field laws making it easier to track interfaces and compute extreme deformations. The employed set of particles represents the state of the system and follows its evolution, being the particles associated with one physical object or part of a continuum representation. The Smoothed Particle Hydrodynamics (SPH) method (Monaghan, 1992; Violeau and Rogers, 2016) is one of the most popular Lagrangian approach, widely used in many water-related fields (Gotth and Khayyer, 2018; Manenti et al., 2019; Amicarelli et al., 2020) and inherently able to address complex fluid-structure interaction in coastal