Numerical Study of K-Semi Motions in Waves Using KRISOFoam

Seongjin Song, Gun Woo Kim, Jae Hwan Jung, Seok Kyu Cho, Young-Myung Choi, Young Jun Kim
Korea Research Institute of Ships and Ocean Engineering, Daejeon, Korea
Department of Naval Architecture and Ocean Engineering, Pusan National University, Busan, Korea

ABSTRACT

A CFD solver KRISOFoam is developed for the simulation of waves and floating body motions using the OpenFOAM package with the waves2Foam library. KRISOFoam is tested for regular and irregular wave simulation. And the motions of K-SEMI (KRISO semi-submersible) in waves are simulated using KRISOFoam.

The validation of KRISOFoam is carried out by analyzing regular and irregular wave simulations. The results are compared with Reproducible Offshore CFD-JIP and are in good agreement. The effects of grid size, aspect ratio, time step, and the number of linear wave superpositions are investigated and compared with CFD-JIP guidelines. It is seen that the smaller the transverse grid size, the better the quality of the waves. Also, the proper number of linear wave superpositions is suggested for irregular wave generation. Adopting the validation results, KRISOFoam is used for K-SEMI motion simulation in waves. It can be seen that KRISOFoam is suitable for simulating waves and floating body motion.

KEY WORDS: KRISOFoam; K-SEMI; OpenFOAM; Irregular wave; Motions

INTRODUCTION

Offshore structures placed in the ocean are exposed to the environments, the ocean waves, wind and current, in their life cycles. For the survival of these structures, the effects of the environments on them must be considered in their design (Chakrabarti, 2005). Among them, many important issues involved in ocean engineering are related to waves. Therefore, the accurate prediction of wave-structure interaction is essential to structure performance and safety in operation. Nowadays, the wave-structure interaction problem can be assessed by various approaches. Empirical formulations, model tests in wave tanks, and numerical simulation methods are all frequently used. The model tests have an advantage of obtaining the accurate and reliable data, but the numerical simulations are relatively advantageous in terms of time and cost. Today, due to improvement and development of powerful computers and computational methods, numerical simulations are mostly used for wave-structure interaction problems.

In numerical simulations, the potential theory-based methods have been often used to predict the hydrodynamic forces and floating structure motions in waves (Chung, 1976; Newman and Lee, 2002). The potential flow solvers are efficient and accurate when the geometry of the structure is smooth and the viscous effects are negligible. However, when the applications involve highly nonlinear waves, wave breaking, strong body nonlinearly, and viscous effect, the potential flow solver can lead to numerical difficulties and be highly ineffective. For this reason, numerous viscous flow solvers based on the Navier-Stokes equations, a viscous CFD solver, have been developed to solve the wave-structure problem.

One of the critical components for wave-structure interaction simulations is the capability to generate waves and propagate them accurately toward the structure. Numerous numerical techniques have been developed to simulate waves for the ocean and civil engineering applications (Windt et al., 2018). Many validation works have been performed to prove that these numerical wave tanks be able to reproduce open sea and experimental conditions for applications of the wave-structure problem (Westphalen et al., 2014; Kim et al., 2016; Gatin et al., 2019; Kim, 2021).

Recently, a CFD modeling practice for a CFD-based Numerical Wave Tank (CNWT) was developed as one of the parts of a joint industry project on Reproducible CFD Modeling Practice for Offshore Applications (Bouscasse et al., 2021). The CFD modeling practice covers the activities performed for verifying and validating regular and irregular waves using two CFD solvers, Star-CCM+ and OpenFOAM. Numerical setups such as computational domain, boundary conditions, turbulence model settings, solver settings, and computational meshes were proposed to inform CFD practitioners who need to simulate wave-structure interaction problems. However, since the recommendations presented are typical, it is necessary to evaluate whether the recommendations are suitable for open-source CFD solvers.

This study uses a CFD solver KRISOFoam to simulate the wave-structure interaction problem under irregular waves. KRISOFoam is developed based on a multiphase solver interDyMFoam in the OpenFOAM package with the wave generation and absorption library waves2Foam. KRISOFoam modified the order of updating weight coefficients within the relaxation zone and the Crank-Nicolson time integration algorithm to improve the efficiency of the code.