The 10th International Conference on Coasts, Ports and Marine Structures (ICOPMAS 2012)
Tehran, Iran, 19-21 Nov. 2012

Numerical modeling of wave- floating body interaction using fictitious domain method

[*Mohammad Javad . Ketabdari*]
[Hassan . Saghi]

***Key Words**: Wave- Floating Body Interaction, Numerical Wave Tank, Volume of Fluid Method, Fictitious Domain Method*

Abstract

In this paper, a 2D numerical model is developed for simulating the interaction between waves and floating body. The fluid flow is assumed to be viscous and incompressible and therefore, Navier–Stokes and continuity equations are used as governing equations. The fast-fictitious-domain method is used into the volume-of-fluid (VOF) technique for tracking the free surface and storage tank motion. The Navier–Stokes equations are discretized using staggered grids finite difference method and solved by SMAC approach. Some standard tests such as lid-driven cavity and constant unidirectional velocity field are performed to valid the model. Finally, floating storage tank motion due to waves is simulated using fictitious domain method. The results show that the maximum vertical floating body motion is increased by about 150% if the wave amplitude is doubled.

Introduction

The free surface deformation in presence of a moving solid object is a complicate phenomenon in the field of fluid dynamics. It is due to complex effects of solid-liquid momentum, the cavity formation behind the solid object and capillary effects at the contact line between the surface of the solid and liquid. Some researchers studied this subject experimentally and theoretically in the past decades. The interaction between a solid body and a liquid medium has a wide application in many engineering problems such as water entry of a solid, ship design, flying boats and sea planes [1]. There are two general approaches to model free surface deformation in presence of a moving solid object. The first category is based on unstructured grids where solid zones are not considered in the computational domain and the object surface are treated as boundary conditions. As the solid body moves inside the fluid the geometry of the fluid computational domain changes. Therefore, a re-meshing is inevitable in each time step. The re-meshing is believed by most researchers as a process that should be avoided. However many studies reported in the literature used this technique in simulation of the fluid-solid interaction. Hu and Joseph used an Eulerian method in which the computational domain is re-meshed in each time step and the solution of the previous time level is mapped into the new mesh. They then solved the Navier-Stokes equations and the implicitly discretized Newton's equations for particle velocities on the new mesh iteratively [2]. Hu adopted an arbitrary Lagrangian-Eulerian