Proceedings of the Tenth (2012) ISOPE Pacific/Asia Offshore Mechanics Symposium
Vladivostok, Russia, October 3-5, 2012

ISBN 978–1 880653 93-7: ISSN 1946-004X
www.isope.org

Numerical Simulation of Violent Evolution of Free Surface during Water Entry of Wedge

Hua Liu[1], Kai Gong[2] and Benlong Wang[1]

1: MOE Key Laboratory of Hydrodynamics, Shanghai Jiao Tong University, Shanghai, China

2: School of Civil and Environmental Engineering, Nanyang Technological University, Singapore

ABSTRACT

A two phase Smoothed Particle Hydrodynamics method is developed to simulate the violent deformation of the free surface and the formation of the cavity induced by the two dimensional wedge entering water. The flow field of water and air are computed simultaneously. A non-reflection boundary treatment for SPH method is proposed to remove reflection of pressure wave against the solid boundaries of computational domain. The water jet generated at the early stage of water entry and the cavity induced by the free surface sealing are reproduced for the case of the vertical water entry of a wedge. The computational pressure distribution and the velocity field can be obtained. Numerical results agree well with the experimental data in literature to verify the two phase SPH method and the boundary treatment approach. It turns out that the cavity evolves with significant deformation of the cavity surface.

KEY WORDS: water entry; SPH; wedge; cavity; two phase model.

INTRODUCTION

Water entry is part of the general fluid-structure impact problem in the field of marine hydrodynamics. Following the pioneer work by von Karman, lots of works have been carried out on hydrodynamic impact on the body entering water from air through the free surface and the evolution of the cavity running with the body. Considering a water entering wedge as a model for studying on slamming, the first complete solution was obtained by Dobrovol'skaya(1969) for a two-dimensional wedge based on potential flow theory and, then, Zhao and Faltinsen(1993) studied theoretically the same problem using a refined procedure. Wu, Sun and He(2004) carried out a numerical and experimental study of this problem more carefully for the early stage of water entry.

The Smoothed Particle Hydrodynamics method (referring as SPH method) is attractive on simulating the violent deformation of the free surface and breaking wave, e.g. Oger et al (2006). It was originally developed for astrophysical computations by Gingold and Monaghan (1977) and has later been extended to model a wide range of problems by Monaghan (1994) and Monaghan and Kocharyan (1995), including multi-phase flow, flow through porous media, heat conduction, and impact and fracture problem. It is a pure Lagrangian, particle method. SPH method does not need a grid to calculate spatial derivatives. Instead, it is based on analytical differentiation of interpolation function. The continuum equation and the momentum conservation equations are formulated as sets of ordinary differential equations. The particle positions and attributes are computed using standard numerical integration methods in time domain such as leap-frog scheme.

For water entry problems, the cavity will appear as an open cavity with atmosphere pressure inside and be enclosed at the later stage of the water entry. In order to obtain cavity profile and hydrodynamic load on the moving body, both water flow and air flow should be considered in mathematical formulation of the problem. Even though there are a lot work on numerical simulation of water entry of a wedge using the potential flow theory and the boundary integral element method, it needs to pay more attention on developing efficient approaches for modeling the violent deformation and sealing of the free surface.(Oger, *et al*. (2004), Gong and Liu (2007))

A two phase SPH model has been implemented to solve flows in water domain and air domain from initial stage of water entry to the closing stage of the cavity, in which water particles and air particles are adopted respectively in these two domains. The mathematical formulation and numerical aspects will be developed and then numerical results of the cavity profile and hydrodynamic loads on the water entering wedge will be discussed.

NUMERICAL MODEL

Mathematical Formulation

The governing equations of fluid dynamics we need to solve are the Navier-Stokes equations. But since we assume the fluid to be non-viscous in the applications presented here, shear stresses are ignored and the equations are reduced to Euler equations as follows: