Proceedings of the Eighth (2008) ISOPE Pacific/Asia Offshore Mechanics Symposium
Bangkok, Thailand, November 10-14, 2008

ISBN 978-1-880653-52-4

Numerical Simulation of Incident Bore Impact Using Three-dimensional Particle Method

Jong-Chun Park[1], *Sung-Jun Jung*[2], *Byung-Hyuk Lee*[1], *Sung-Chul Hwang*[1]
[1] Dept. of Naval Architecture and Ocean Engineering, Pusan National Univ.
Busan, Korea
[2] Ship and Ocean R&D Institute, Daewoo Shipbuilding and Maritime Engineering Co. Ltd.
Geoje, Korea

ABSTRACT

The impact of a single wave generated by a dam break with a tall structure is modeled with a three-dimensional version of the Moving particle semi-implicit (MPS) method. The particle method is more feasible and effective than methods based on grid connection problems involving the violent free surface motions. In the present study, the incident bore impact load and the change of longitudinal velocity component around the structure, which are obtained from the numerical simulation, are compared those from experiments by Arnason (2005).

KEY WORDS: Particle method; Moving particle semi-implicit (MPS); Non-linear free-surface motion; Bore impact load

INTRODUCTION

The accurate prediction of impact loads by fluid gives important information for safety of ships or maritime structures. The large deformation and dynamic behavior of free surface are one of the most difficult problems for numerical simulations because the numerical implementation of the fully nonlinear free-surface condition is in general complicated and difficult. There are several techniques to handle such kinds of problems, i.e. SOLA-VOF (Hirt and Nichols, 1981), Level-Set (Sussman et al., 1994), Marker-Density function (MDF) (Miyata and Park, 1995) etc.. Most of them are the techniques capturing the free-surface on grid system. However, there is a different approach using no-grid system, so-called particle methods by use of moving particles with the Lagrangian treatment. The particle methods seem to be more feasible and effective than conventional grid methods for solving the flow field with complicated boundary shapes or the coupling effects between fluid and structure.

In the present study, the violent free-surface motions interacting with structures are investigated using the Moving Particle Semi-implicit (MPS) method, which was proposed by Koshizuka and Oka (1996) for incompressible flow. In the method, Lagrangian moving particles are used for solving flow field instead of Eulerian approach using grid system. Therefore convection term in Navier-Stokes equation can be directly calculated without numerical diffusion or instabilities due to the fully Lagrangian treatment of particles and topological failure never occur. The method consists of the particle interaction models representing gradient, diffusion, incompressibility and the boundary conditions.

The simulated bore impact load and the change of longitudinal velocity component around the structure are compared to those from experiments. Those simulation condition and other numerical results were referred from Raad and Bioae (2005), Raad(website) and Lin et al. (2005).

GOVERNING EQUATIONS

The Governing equations for incompressible viscous flows are the continuity and Navier-Stokes equations as follows:

$$\frac{D\rho}{Dt} = 0 \tag{1}$$

$$\frac{D\vec{u}}{Dt} = -\frac{1}{\rho}\nabla P + \nu\nabla^2\vec{u} + \vec{F} \tag{2}$$

The symbol ρ is the density, t the time, $\vec{u}$ the velocity vector, ∇ the gradient, P the pressure, ν the kinematic viscosity, and $\vec{F}$ the external force.

The continuity equation (Eq. 1) is written with respect to the density, while velocity divergence is usually used in grid methods. The left-hand side of Navier-Stokes equation (Eq. 2) denotes Lagrangian differentiation including convection terms. This is directly calculated by moving particles. The right-hand sides consist of pressure gradient, viscous, and external-force terms. To simulate incompressible flows, all terms expressed by differential operators should be replaced by the particle interaction models of the MPS method.

PARTICLE INTERACTION MODEL

Kernel Function

Continuous fluid can be represented by physical quantities of coordinates, mass, velocity components, and pressure for particles. The governing equations written with partial differentiations are transformed to the equation of particle interactions. Particle interactions in the MPS method are based on the kernel function. In this study, the following function is employed:

$$w(r) = \begin{cases} \dfrac{r_e}{r} - 1 & (0 \le r < r_e) \\ 0 & (r_e < r) \end{cases} \tag{3}$$

The distance between two particles is r and r_e represents the effective