Ocean Engineering 69 (2013) 70–78

Contents lists available at SciVerse ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Numerical prediction of oil amount leaked from a damaged tank using two-dimensional moving particle simulation method

Se-Min Jeong [a], Jung-Woo Nam [a], Sung-Chul Hwang [a], Jong-Chun Park [a,*], Moo-Hyun Kim [b]

[a] Department of Naval Architecture and Ocean Engineering, Pusan National University, Busan, Republic of Korea
[b] Department of Civil Engineering, Texas A&M University, TX, USA

A R T I C L E I N F O

Article history:
Received 28 December 2012
Accepted 11 May 2013
Available online 10 June 2013

Keywords:
Moving particle simulation method
Oil leaking
Torricell's euilibrium equation
Gradient model
High order time integration

A B S T R A C T

When an oil-spilling accident occurs at sea, it causes tremendous economic loss and environmental damages. To minimize these marine disasters, predicting the amount of oil-leaking is one of the most important things for the quick response and decision making in the early stage of the accident. In the present study, numerical investigation on the oil-leaking phenomena from a two-dimensional damaged tank was carried out to predict the amount of the oil leaking from a leakage hole of a rectangular tank by using the PNU-MPS(Pusan-National-University-modified Moving Particle Simulation) method (Lee et al., 2011). As a preliminary test to confirm the accuracy of the PNU-MPS method for the two-phase problem and to investigate the influence of the gradient model, numerical simulations for the Rayleigh–Taylor(R–T) instability were carried out By using an appropriate gradient model and applying a high-order time integration scheme, i.e. 4th-order Runge–Kutta scheme, it was found that the simulation results became closer to the experimental ones (Kim and Lee, 2001), by which the oil-leaking speed and Torricelli's factor relating the speed predicted by using the hydrostatic balance and the real leakage speed were measured and assessed.

1. Introduction

Marine oil spills, in which oil is released into the ocean or coastal waters, are mainly due to the releases of crude oil from tankers, offshore platforms, drilling rigs and wells, as well as spills of refined petroleum products and their by-products, heavier fuels used by large ships such as bunker fuel, or the spill of any oily refuse or waste oil. When an oil-spilling accident occurs at sea, such as Exxon Valdez oil spill (1989) and oil spill of the west coast of Korea (2007), it causes tremendous economic loss and environmental damages. To minimize these marine disasters, predicting the trajectory and amount of oil-leaking are important for the quick response and decision making in the early stage of the accident.

To predict the amount of oil-leaking, Torricelli's equilibrium equation is convenient and useful although it is very simple (Simecek-Beatty et al., 2001). However, as the equation does not consider the influence of the viscosity, the shape of an opening, interaction with external fluid in the vicinity of outlet and so forth, precise evaluation cannot be acquired by this equation. Kim and Lee (2001) performed both experimental and numerical studies to find out the oil-leaking speed and its dependency on the shape of leakage hole. From the experiments, it was found that the oil-leaking speed is dependent of aspect ratio of leakage holes even though they are of the same area. And, the Torricelli's factor relating the speed predicted by using the hydrostatic balance and the real leakage speed was assessed. Recently, Tavakoli et al. (2012) developed analytic models for estimation of oil spill from a side tank with collision damage and performed CFD (Computational Fluid Dynamics) simulation with Fluent software for verification of the simple models.

Although most of the simulations for the oil-leaking phenomena are based on the Eulerain grid based method using interface capturing methods, such as SOLA-VOF (Volume Of Fluid) (Hirt and Nichols, 1981), Level-Set (Sussman et al., 1994), Marker-Density Function (MDF) (Miyata and Park, 1995) and so on, there is a different approach without using a grid system, which is called the particle method by using the moving particles in the Lagrangian frame, such as SPH (Smoothed Particle Hydrodynamics) (Gingold and Monaghan, 1977) and MPS (Moving Particle Semi-implicit/Simulation) (Koshizuka et al., 1996).

In the present study, numerical studies on the oil-leaking phenomena from a two-dimensional damaged tank were carried out to predict the amount of the oil leaking from a leakage hole of the tank by using the PNU-MPS (Pusan-National-University-modified MPS) method (Lee et al., 2011), which is an improved version of original MPS (Koshizuka et al., 1996).

* Corresponding author. Tel.: +82 51 510 2480; fax: +82 51 583 2485.
E-mail address: jcpark@pnu.edu (J.-C. Park).

0029-8018/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.oceaneng.2013.05.009

2. Moving particle simulation (MPS) method

2.1. Governing equations

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

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

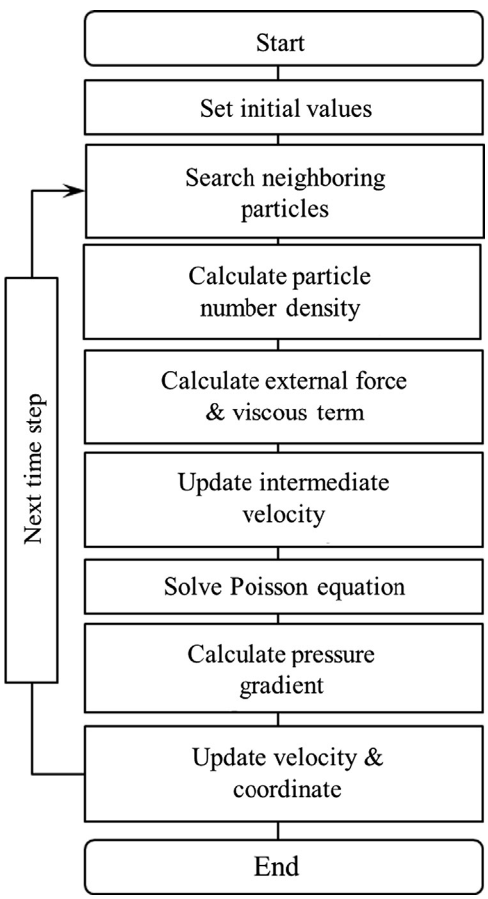

Fig. 1. Computational procedure of PNU-MPS.

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

Here 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 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 Eq. (2) denotes Lagrangian differentiation that is directly calculated by moving particles in a Lagrangian manner. The right-hand side consists 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 proposed originally by Koshizuka et al. (1996). In the present paper, the refined MPS (called PNU-MPS form this point on) method (Lee et al., 2011) is employed for all simulations, which can calculate the flow field with violent free-surface motion more accurately and stably compared with the original MPS (Koshizuka et al., 1996).

2.2. 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 differential operators are transformed to the equation of particle interactions. The particle interactions in the MPS method are based on the kernel function. In this study, the following function is employed

$$w(r)=\begin{cases}\left(1-\frac{r}{r_e}\right)^3\left(1+\frac{r}{r_e}\right)^3 & (0\le r<r_e)\\ 0 & (r_e<r)\end{cases} \tag{3}$$

The distance between two particles is r and $r_e$represents the effective range of particle interactions. The kernel (3) seems to be more robust and gives more reasonable interaction between neighboring particles for the weakly-violent free-surface problem than that used in the original MPS method as Lee et al. (2011) pointed out.

2.3. Gradient model

Although a gradient vector between two particles i and j possessing scalar quantities ϕ_i and ϕ_j at coordinates r_i and r_j is defined as $(\phi_j-\phi_i)(\vec{r_j}-\vec{r_i})/|\vec{r_j}-\vec{r_i}|^2$ in the original MPS method (Koshizuka et al., 1996), the PNU-MPS method adopts the gradient

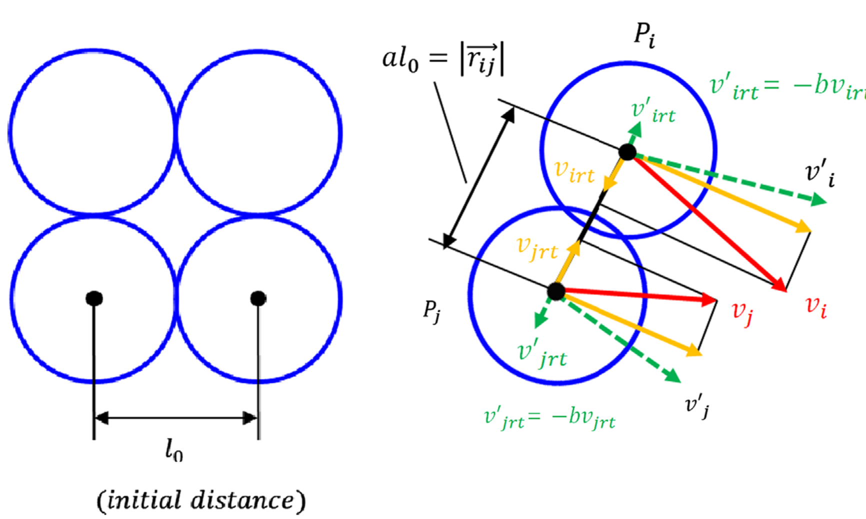

Fig. 2. Diagram of collision model.