Available online at www.sciencedirect.com

ScienceDirect

Journal of Hydro-environment Research 9 (2015) 340–353

Journal of Hydro-environment Research

www.elsevier.com/locate/jher

Research paper

Incompressible SPH simulation of open channel flow over smooth bed

Soon Keat Tan [a], Nian-Sheng Cheng [b], Yin Xie [a], Songdong Shao [c,d,*]

[a] NEWRI and Maritime Research Centre, Nanyang Technological University, Singapore 639798, Singapore
[b] School of Civil and Environmental Engineering, Nanyang Technological University, Singapore 639798, Singapore
[c] Department of Civil and Structural Engineering, University of Sheffield, Sheffield S1 3JD, United Kingdom
[d] School of Water Resources and Electric Power, Qinghai University, Xining 810016, China

Received 17 May 2014; revised 30 October 2014; accepted 22 December 2014
Available online 28 May 2015

Abstract

The Smoothed Particle Hydrodynamics (SPH) modelling techniques are used in a variety of coastal hydrodynamic applications, but only limited works have been documented in the open channel flows. In this paper, we use the incompressible SPH model combined with an improved inflow boundary scheme to investigate the open channel flows in a laboratory scale. The inflow and outflow boundary conditions are treated by generating and removing the fluid particles at the channel end. The proposed ISPH model has been applied to the open channel laminar and turbulent flows of different flow depths and the computational results have been verified against the analytical solutions. Model convergence has been investigated through the numerical tests using different particle spacings and time steps. The artificial boundary drag force of numerical nature has been found under certain flow conditions. As further application of the model, two additional tests are also carried out, involving alternative solid boundary treatment and more complex channel topography. The present study could provide useful information on further exploitation of the SPH modelling technique in river hydrodynamics.

Keywords: ISPH; Inflow boundary; Open channel flow; Artificial drag force; Smooth bed

1. Introduction

The Smoothed Particle Hydrodynamics (SPH) method is a pure Lagrangian modelling technique for the free surface flows. However, most SPH applications have been documented in the coastal hydrodynamics, such as wave breaking (Monaghan and Kos, 1999; Dalrymple and Rogers, 2006), wave overtopping (Gomez-Gesteira et al., 2005) and wave interaction with breakwater (Rogers et al., 2010). Other areas of common SPH application are related to some rapid unsteady flows, such as dam break (Afshar and Shobeyri, 2010), water entry of an object (Oger et al., 2006) and landslide-induced wave (Ataie-Ashtiani and Shobeyri, 2008). It has been noted that very few SPH works are reported in the open channel flows. One reason is that the simulation of open channel flow in a steady state requires much more CPU time as compared with other SPH simulations. Another reason is the SPH treatment of inflow and outflow boundaries. In the SPH framework the particles need to be generated and removed on the flow boundaries and thus a strict satisfaction of the boundary condition is difficult to achieve. Besides, the interpolation procedures of SPH numerical nature make the implementation of this kind of boundary condition quite challenging.

In spite of this, some promising progresses have been made in recent years in the SPH application under inflow and outflow conditions. For example, Lee et al. (2008) first proposed a periodic inflow/outflow boundary for the open channel

* Corresponding author. Department of Civil and Structural Engineering, University of Sheffield, Sheffield S1 3JD, United Kingdom.

E-mail addresses: ctansk@ntu.edu.sg (S.K. Tan), cnscheng@ntu.edu.sg (N.-S. Cheng), yxie1987@gmail.com (Y. Xie), s.shao@sheffield.ac.uk (S. Shao).

http://dx.doi.org/10.1016/j.jher.2014.12.006
1570-6443/

laminar flow around a bluff body by using the weakly compressible/incompressible SPHs. Later Lastiwka et al. (2009) developed a WCSPH model for the imposition of permeable boundary conditions for the gas dynamics. Moreover, Shakibaeinia and Jin (2010) and Federico et al. (2012) used different particle recycling techniques for more practical open channel hydraulics. Gotoh et al. (2001) developed a novel soluble wall concept to generate the inflows and simulated a free turbulent jet by using the Moving Particle Semi-implicit (MPS) model. More comprehensive evaluations of these pioneering works on their pros and cons will be summarized in the later section on Inflow and Outflow Boundaries.

This paper is structured as follows. In the next section the fundamental principles of ISPH model are reviewed, followed by a comprehensive evaluation of existing inflow/outflow boundary works in particle-based models and the development of an improved inflow ISPH numerical scheme to be used in the open channel flows. In the model applications, we first simulate two laminar flows and three turbulent flows with different flow depths and verify the velocity profiles by the analytical solutions. Then a series of numerical analyses are carried out to examine the influence of artificial boundary drag forces arising from the SPH solid boundary treatment. Moreover, model convergence in the spatial and temporal domains is investigated by using different particle spacings and time steps. Finally, two more critical model tests are made to improve the near-wall simulation accuracy and investigate the pressure stability on the inlet boundary under complex channel bed configurations.

2. Principles of ISPH model

2.1. Governing equations

The ISPH model solves the Navier–Stokes (NS) equations in Lagrangian form as

$$\frac{1}{\rho}\frac{d\rho}{dt} + \nabla\cdot\mathbf{u} = 0 \tag{1}$$

$$\frac{d\mathbf{u}}{dt} = -\frac{1}{\rho}\nabla P + \mathbf{g} + \nu_0\nabla^2\mathbf{u} + \frac{1}{\rho}\nabla\cdot\vec{\vec{\tau}} \tag{2}$$

where ρ = fluid particle density; t = time; $\mathbf{u}$ = particle velocity vector; P = particle pressure; $\mathbf{g}$ = gravitational acceleration vector; ν_0 = laminar kinematic viscosity; and $\vec{\vec{\tau}}$ = turbulent stress. It should be realized that quite a few compressible flows have been directly solved by SPH using the above equations. However, we found that the incompressible flow model was more effective in terms of reducing the particle fluctuation and pressure noises, so that it can better treat the inflow and outflow boundaries in an open channel flow. In the present model, incompressibility of the fluid is imposed by the density invariant condition on each of the inner fluid particles.

The turbulent stress $\vec{\vec{\tau}}$ in Equation (2) should be modelled in open channel turbulent flows, as the computational particle scale is much larger than the flow turbulent structure. By following a simple and effective eddy viscosity based Sub-particle Scale (SPS) turbulence formulation, which was originally proposed by Gotoh et al. (2001) for a turbulent jet, we have

$$\tau_{ij}/\rho = 2\nu_T S_{ij} - \frac{2}{3}k\delta_{ij} \tag{3}$$

where ν_T = turbulent eddy viscosity; S_{ij} = strain rate of mean flow; k = turbulent kinetic energy; and δ_{ij} = Kronecker's delta. Here the following Smagorinsky model is used to compute the turbulent eddy viscosity ν_T as follows:

$$\nu_T = (C_s\Delta X)^2|S| \tag{4}$$

where C_s = Smagorinsky constant, which is taken 0.1 in this paper; ΔX = particle spacing, which represents the characteristic length scale of the small eddies; and $|S| = (2S_{ij}S_{ij})^{1/2}$ is the local strain rate.

2.2. Solution procedures

The ISPH's prediction and correction solution scheme consists of two steps. The prediction step is an explicit integration in time without enforcing the incompressibility, during which only the gravitational, viscous and turbulent forces in Equation (2) are used and then an intermediate particle velocity and position are obtained as

$$\Delta\mathbf{u}_* = \left(\mathbf{g} + \nu_0\nabla^2\mathbf{u} + \frac{1}{\rho}\nabla\cdot\vec{\vec{\tau}}\right)\Delta t \tag{5}$$

$$\mathbf{u}_* = \mathbf{u}_t + \Delta\mathbf{u}_* \tag{6}$$

$$\mathbf{r}_* = \mathbf{r}_t + \mathbf{u}_*\Delta t \tag{7}$$

where $\Delta\mathbf{u}_*$ = changed particle velocity vector at the prediction step; Δt = time increment; $\mathbf{u}_t$ and $\mathbf{r}_t$ = particle velocity and position vectors at time t; and $\mathbf{u}_*$ and $\mathbf{r}_*$ = intermediate particle velocity and position vectors.

The correction step is to modify the density of fluid particles to initial values and the pressure term is used to update the particle velocity obtained from the intermediate step as

$$\Delta\mathbf{u}_{**} = -\frac{1}{\rho_*}\nabla P_{t+1}\Delta t \tag{8}$$

$$\mathbf{u}_{t+1} = \mathbf{u}_* + \Delta\mathbf{u}_{**} \tag{9}$$

where $\Delta\mathbf{u}_{**}$ = changed particle velocity vector at the correction step; ρ_* = intermediate particle density between the prediction and correction steps; and P_{t+1} and $\mathbf{u}_{t+1}$ = particle pressure and velocity vector at time $t+1$. The final positions of a particle are centred in time as

$$\mathbf{r}_{t+1} = \mathbf{r}_t + \frac{(\mathbf{u}_t + \mathbf{u}_{t+1})}{2}\Delta t \tag{10}$$

where $\mathbf{r}_t$ and $\mathbf{r}_{t+1}$ = position vectors of the particle at time t and $t+1$, respectively.