Ocean Engineering 105 (2015) 160–175

Contents lists available at ScienceDirect

Ocean Engineering

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

# SPH modeling of plane jets into water bodies through an inflow/outflow algorithm

Francesco Aristodemo [a,*], Salvatore Marrone [b], Ivan Federico [c]

[a] Facoltá di Ingegneria, Universitá degli Studi eCampus, via Isimbardi 10, 22060 Novedrate (CO), Italy
[b] CNR-INSEAN, The Italian Ship Model Basin, Via di Vallerano 139, 00128 Roma, Italy
[c] OceanLab, CMCC, Euro-Mediterranean Centre on Climate Change, Via Augusto Imperatore 16, 73100 Lecce, Italy

ARTICLE INFO

*Article history:*
Received 29 May 2014
Accepted 15 June 2015
Available online 6 July 2015

*Keywords:*
Smoothed Particle Hydrodynamics
Inflow/outflow boundary conditions
Jets
Water tanks
Channel flows

ABSTRACT

This paper deals with the development and application of a two-dimensional weakly compressible Smoothed Particle Hydrodynamics (SPH) model to study plane jets propagating into still fluid tanks and current flows. These flow processes occurring in different water bodies are here treated through an appropriate algorithm to model inlet/outlet boundary conditions which are defined by different sets of particles. SPH equations of fluid mechanics including viscous and interface stabilization terms are adopted to determine the flow field induced by the fluid interaction. Flow phenomena induced by jet injection in a water body occur in natural and anthropic environments such as pollutant discharge into reservoirs, rivers and coasts, or in several engineering applications such as marine outfalls. Three test cases with different initial configurations are simulated: injection of a jet at the top and at the bottom of a water tank, and upstream of a channel flow (coflow jet). The proposed SPH model is validated both near the jet nozzle and far from it, comparing the obtained jet trajectories, widths and velocities with other numerical approaches, laboratory experiments and analytical solutions.



## 1. Introduction

The use of meshless methods for Computational Fluid Dynamics (CFD) has experienced an exponential growth during the last decade (e.g., Koumoutsakos, 2005; Liu and Liu, 2010). These methods, whose main idea is to substitute the grid with a set of arbitrarily distributed nodes, are expected to be more adaptable and versatile than the conventional grid-based methods, especially for those applications with severe discontinuities in the fluid. In particular, particle meshfree methods have proved to be well suited for simulating free surface flows and interactions between solids and the water with sharp changes of pressure and velocity in the flow field, including complex phenomena such as bores and splashes (e.g., Landrini et al., 2007).

Smoothed Particle Hydrodynamics (SPH) is a meshless Lagrangian method which proves to be well suited for simulating in detail complex fluid dynamics. First applied to astrophysical problems (Gingold and Monaghan, 1977), this method has been successfully used to model free-surface flows (e.g., Monaghan, 1994) especially when strong free-surface deformations take place, such as interfacial flows (e.g., Colagrossi and Landrini, 2003; Grenier et al., 2009), dam-breaks (e.g., Aristodemo et al., 2008), fluid–structure interactions (e.g., Marrone et al., 2011, 2013; Aristodemo et al., 2015), sloshing flows (e.g., Delorme et al., 2009) and advective diffusion processes induced by pollutants in water (Aristodemo et al., 2010). Following the SPH method, the motion of a continuum medium is described using an interpolation technique which allows the approximation of functions and differential operators on an irregular distribution of points.

With reference to the Eulerian numerical approaches, the enforcement of inlet/outlet boundary conditions is quite straightforward because each cell of the mesh describes a part of the computational domain and ghost cells can be adopted to impose boundary conditions. As is common knowledge, the enforcement of these conditions is not trivial using particle models such as SPH model due to the Lagrangian nature and the interpolation procedure at the basis of its numerical scheme. Gomme et al. (2006) modeled the motion of a lobe pump using a periodic boundary condition between the inlet and the outlet. Lastiwka et al. (2009) developed an SPH model through the imposition of permeable boundary conditions to simulate uniform flows for gas dynamic problems but without any modeling of the free surface. More recently, Federico et al. (2012) have proposed a more general SPH-based algorithm to model inflow/outflow boundary conditions for treating free-surface channel flows with application to uniform, non-uniform and unsteady flow regimes which can occur in water streams. Through the assumption of suitable inflow and outflow buffer particles, this algorithm allows the enforcement of different inlet/outlet boundary conditions. This inflow/outflow algorithm has also been successfully applied to model

* Corresponding author. Tel.: +39 0984 496554; fax: +39 0984 494050.
*E-mail address:* francesco.aristodemo@uniecampus.it (F. Aristodemo).

http://dx.doi.org/10.1016/j.oceaneng.2015.06.018
0029-8018/

a broad range of flow processes such as the interaction between free-surface steady currents and boats (Marrone et al., 2012) or bluff bodies (Marrone et al., 2013), gated spillway flows (Saunders et al., 2014), flow separation at bends (Hou et al., 2014) and flow over sills and weirs (Meister et al., 2014). In this context, the modeling of jets discharged into reservoirs and channel flow through the SPH technique needs appropriate inlet/outlet boundary conditions. Until now, limited SPH simulations have been carried out to study these physical processes and previous works (Espa et al., 2008; Sibilla and Torti, 2012) did not provide the conditions to develop a general and suitable numerical approach to analyze a broad range of jets evolving in static and dynamic fluid bodies.

Jets have been investigated for many years by the fluid mechanics community (e.g., Fischer et al., 1979; Wood et al., 1993). The interest comes from the importance of these phenomena in several environmental flows. Jet discharges from industrial and natural sources often enter fluvial, coastal and offshore areas. This flow configuration is of theoretical significance in environmental fluid mechanics due to the complex interaction between jet and ambient flow. This interaction can lead to vortical structures which play a fundamental role in the entrainment of the ambient fluid into the discharge jet (Jirka, 2004).

Here, the basis of the computational method given by Federico et al. (2012) is extended to model the dynamics of fluid processes characterized by a time-constant injection of a tracer into still fluid tanks and channel flows. In order to determine the main flow phenomena induced by jet–water interaction in a confined region, appropriate inflow particles are introduced at different locations to simulate a jet insertion in a static or dynamic water ambient as well as other sets of particles are adopted to model the presence of fluid and solid boundaries. Initial velocities and pressures, both inflow and outflow in the computational domain, are defined.

Firstly, attention is paid to the study of two-dimensional jets propagating into water tanks, performing the analysis in shallow water environments, characterized by a low ratio between the water depth and the jet diameter (e.g., Andreopoulos et al., 1986). Successively, the evolution of a plane jet in the presence of a coexisting flow, as in the case of a channel flow, is considered. The adopted fluid mechanics equations prove to be suitable for modeling the induced flow field by considering fluids having the same density. SPH simulations are performed in near field (close to the jet nozzle) and far ones.

In the following section the bases of the SPH approach and the adopted governing equations are illustrated. Afterwards, the algorithm for modeling the discharge of a jet into a surrounding static or dynamic fluid environment through appropriate boundary conditions is described as well as the treatment of other boundaries and the computational strategies are illustrated. Comparisons of the proposed SPH model with other numerical approaches, experimental data and analytical solutions are finally reported, showing the evolution of jet shape (penetration and width) and their associated velocity field given by the flow interaction.

## 2. SPH scheme

### 2.1. Main characteristics

The Lagrangian meshless approach follows the idea that the fluid can be modeled as a finite set of particles, each one with its local mass and other physical properties. Following the evolution of those particles, Partial Differential Equations are solved considering the information they carry to reconstruct the fluid properties everywhere in the domain. In the equations of fluid dynamics, the rates of change of physical quantities require spatial derivatives of physical quantities. The key step in any computational fluid dynamics algorithm is to approximate these derivatives using information from a finite number of points. In the Smoothed Particle Hydrodynamic method, the interpolating points are particles which move with the flow, and the interpolation of any quantity, at any point in space, is based on kernel estimation (Monaghan, 2005).

SPH interpolation of the variable $f$, which is a function of the spatial coordinates, is based on the integral interpolant

$$\langle f(\mathbf{r})\rangle = \int_{\Omega} f(\mathbf{r}')W(\mathbf{r}-\mathbf{r}';\epsilon)\,dV', \tag{1}$$

where $\mathbf{r}$ is the position and $f$ is evaluated by interpolating its known values in $\mathbf{r}'$ over the domain $\Omega$. In Eq. (1) $W(\mathbf{r}-\mathbf{r}';\epsilon)$ is a weight function and $\epsilon$ is a measure of the support of $W$, i.e., where $W$ is other than zero. Physically, $\epsilon$ is also representative of the domain of influence $\Omega_{\mathbf{r}'}$ of $\mathbf{r}'$. In the SPH framework, $W(\mathbf{r}-\mathbf{r}')$ is called *smoothing function* or *kernel* and has several properties (see Monaghan, 2005 for more details). For $\epsilon \to 0$, the kernel function $W$ becomes a Dirac delta function $\delta$.

The approximations of the derivatives of the field $f$ can be expressed through the derivatives of the kernel as

$$\langle \nabla f(\mathbf{r}')\rangle = \int_{\Omega} f(\mathbf{r}')\nabla W(\mathbf{r}-\mathbf{r}';\epsilon)\,dV'. \tag{2}$$

The value of $f$ at the reference particle $a$ is denoted by $f_a$ with associated mass $m_a$, density $\rho_a$ and position $\mathbf{r}_a$. Eq (1) can be approximated by the following summation interpolant over the mass elements:

$$\langle f_a\rangle = \sum_b \frac{f_b}{\rho_b} m_b W(\mathbf{r}_{ab}) \tag{3}$$

where $W(\mathbf{r}_{ab}) = W(\mathbf{r}_a-\mathbf{r}_b;\epsilon)$ represents the kernel centered at the $b$-th particle position and evaluated at the $a$-th particle position. The subscript $b$ refers to one of the other neighbor particles in the domain.

The SPH formulation allows derivatives to be estimated as

$$\langle \nabla f(\mathbf{r}_a)\rangle = \frac{1}{\rho_a}\sum_b m_b (f_b - f_a)\nabla_a W(\mathbf{r}_{ab}) \tag{4}$$

Eqs. (3) and (4) represent the basic discrete formulations of the SPH method.

### 2.2. Governing equations

The reference equations for the global flow evolution assuming a viscous, weakly compressible and barotropic fluid are

$$\begin{cases} \dfrac{D\mathbf{v}}{Dt} = -\dfrac{1}{\rho}\nabla p + \mathbf{g} + \dfrac{1}{\rho}\nabla\cdot\mathbf{V} \\ \dfrac{D\rho}{Dt} = -\rho\nabla\cdot\mathbf{v} \\ p = c_0^2(\rho-\rho_0) \\ \dfrac{D\mathbf{r}}{Dt} = \mathbf{u} \end{cases} \tag{5}$$

where $\mathbf{v}$, $p$ and $\rho$ are, respectively, velocity, pressure and density of a generic material point, $\mathbf{g}$ represents the mass force acting on the fluid, $\rho_0$ the initial density, $c_0$ the initial sound speed and $\mathbf{V}$ the viscous stress tensor. The weak-compressibility is enforced by using a linearized version of Tait state equation which relates pressure and density (e.g., Molteni and Colagrossi, 2009).

The use of a weakly compressible SPH model allows the avoidance of the adoption of the typical value of the physical sound velocity but, conversely, the value of sound speed, $c$, which can be obtained through the following constraint:

$$c = \max\{10\max(|\mathbf{u}|), 10\sqrt{gH}\} \tag{6}$$

where $H$ is the water depth and the characteristic velocity, $\mathbf{u}$, is here set equal to the initial jet velocity. Although the physical sound