Journal of Hydrology 541 (2016) 344–358

Contents lists available at ScienceDirect

Journal of Hydrology

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

Modelling large floating bodies in urban area flash-floods via a Smoothed Particle Hydrodynamics model

Raffaele Albano [a,*], Aurelia Sole [a], Domenica Mirauda [a], Jan Adamowski [b]

[a] School of Engineering, University of Basilicata, Italy
[b] Department of Bioresource Engineering, McGill University, Canada

ARTICLE INFO

Article history:
Available online 13 February 2016

Keywords:
Smoothed Particle Hydrodynamics
Floating bodies transport
Flash-flood
Structural mitigation action
Lagrangian modelling
Fluid–structure interaction

SUMMARY

Large debris, including vehicles parked along floodplains, can cause severe damage and significant loss of life during urban area flash-floods. In this study, the authors validated and applied the Smoothed Particle Hydrodynamics (SPH) model, developed in Amicarelli et al. (2015), which reproduces in 3D the dynamics of rigid bodies driven by free surface flows, to the design of flood mitigation measures. To validate the model, the authors compared the model's predictions to the results of an experimental setup, involving a dam breach that strikes two fixed obstacles and three transportable floating bodies. Given the accuracy of the results, in terms of water depth over time and the time history of the bodies' movements, the SPH model explored in this study was used to analyse the mitigation efficiency of a proposed structural intervention – the use of small barriers (groynes) to prevent the transport of floating bodies. Different groynes configurations were examined to identify the most appropriate design and layout for urban area flash-flood damage mitigation. The authors found that groynes positioned upstream and downstream of each floating body can be effective as a risk mitigation measure for damage resulting from their movement.

1. Introduction

Many obstacles may be present in rivers during urban flood events, in both the main channel (bridge piers) and on the floodplains (e.g. abutments, dikes, trees and vegetation, and debris from former floods). Moreover, if river embankments are overcome, or flood dikes breached, the flow will enter areas that are normally not subject to inundation and are therefore not prepared to support such events. These areas generally contain a multitude of other obstacles: roads, vehicles, railways, dwellings, and industrial and commercial structures. The presence of these artificial obstacles can considerably affect water flows. Furthermore, in the event of severe and rapid floods (e.g. flash-floods), the influence of such obstacles is amplified, especially in urban environments. Large debris flow can clog natural sections, forming "debris dams" or "valley jams." In turn, debris dams can contribute to increased water levels up-stream and can cause dam-break flows in the event a sudden breach of large impounded water volumes. These potential impacts are similar to the impacts of temporary dams created by landslide deposits (Lucìa et al., 2015).

Large floating debris transported by the floodwater flow can significantly worsen the effects of flooding by blocking fluvial and road infrastructures and increasing water levels. For example, during the Boscastle flood in England in August 2004, 115 vehicles were swept away by floodwater; some of these vehicles were caught under a local bridge, thereby blocking the flow path and ultimately contributing to the collapse of the bridge due to stress (Teo et al., 2012). Furthermore, during the Rapid City flash flood in South Dakota, USA, on June 9, 1972, 38 cm of precipitation accumulated in less than six hours. As a result, Rapid Creek rose 3.66 m after the spillway at Pactola Dam, located upstream of Rapid Creek, became blocked with cars and house debris (Gruntfest and Ripps, 2000). Large debris, including vehicles parked along floodplains, can cause severe damage and significant loss of life. For example, a high rate of mortality associated with vehicles and floods was documented in the 1976 Big Thompson flood in Colorado, USA (Gruntfest, 1997, 2000).

Computational Fluid Dynamics (CFD) modelling for urban flood events is very useful for characterizing the complexity of an urban system. It also allows for a detailed evaluation and understanding of phenomena such as the transport of solid structures (e.g. vehicles and tree trunks). Accordingly, CFD enables one to perform numerical experiments, rather than expensive and, in some cases, impossible physical experiments, where similarity principles cannot be invoked and scale models cannot be used (Violeau, 2012).

---

* Corresponding author.
E-mail address: albano.raffaele@tiscali.it (R. Albano).

http://dx.doi.org/10.1016/j.jhydrol.2016.02.009
0022-1694/© 2016 Elsevier B.V. All rights reserved.

CFD modelling also provides additional information that cannot be obtained from direct experimental observation. This is particularly valuable when the aim of the study is not only to describe a flow variable, but rather to understand the physical process controlling the phenomena (Violeau, 2012).

The use of mesh-free methods for CFD has grown exponentially during the last decade. These methods, whose main idea is to substitute the grid by a set of arbitrarily distributed nodes, are expected to be more adaptable and versatile than the conventional grid-based approaches, especially for those applications with severe discontinuities in the fluid. In this context, the Smoothed Particle Hydrodynamics (SPH) model represents a mesh-less CFD technique to simulate free surface and flow impact on fixed and mobile structures and multi-phase flow modelling (Monaghan, 2005; Liu et al., 2013; Vaughan, 2009). It is particularly appropriate for the representation of dynamical flood events, such as urban floods involving obstacles or dam-break conditions, both in terms of their dynamics and with regards to the forecast of their effects (Viccione and Bovolin, 2011).

The main advantages of this technique concern (1) the direct estimation of the free-surface and the interfaces between fluids or phases, as defined by the positions of the numerical particles; (2) the effective management of multiple moving bodies or the transported particle matter; and (3) the computation of Lagrangian parameters and derivatives, avoiding the direct treatment of the non-linear advective term in the Navier–Stokes equation (Gómez-Gesteira and Dalrymple, 2004; Liu and Liu, 2003). Furthermore, the algorithm is rather simple when compared to Eulerian modelling since it requires neither iterative convergence procedures, nor a computational mesh. In particular, this technique provides the greatest advantage when applied to fast flows in transitory regimes. On the other hand, SPH is generally more time-consuming than Eulerian CFD techniques since the numerical stencil of each computational node is composed of approximately one hundred particles in 3D, rather than a tenth of cells for mesh-based models (Viccione et al., 2008). Still, the algorithm is appropriate for parallelization, noticeably reducing the negative effects of this shortcoming (Gomez-Gesteira et al., 2010; Violeau, 2012). Conceptually, the method uses integral theory to transform the partial differential equations into an integral form. Moreover, the SPH approach can simultaneously deal with multiple body dynamics, as are usually developed in astrophysics and solid mechanics (Monaghan, 2005; Liu and Liu, 2003; Violeau, 2012). To date, only a few SPH models have been developed to represent the transport of moving bodies driven by 3D free surface flows. The main difficulties stem from the treatment of each of the multiple 2-way fluid–body and solid–solid (body–body and body–boundary) interactions.

Monaghan et al. (2003) described a SPH numerical method based on boundary force particles in order to model the impact and entry of a rigid body travelling down a slope into water. Using the same technique, Omidvar et al. (2012a,b) investigated the impact of a float device in free-surface waves. The reliability of the boundary force particles technique in reproducing 2D modular bodies in confined flows was highlighted by Kajtar and Monaghan (2010) and later coupled with repulsive forces to model the body force owing to rigid boundaries, (Kajtar and Monaghan, 2012). Hashemi et al. (2011) simulated 2D moving solid bodies in visco-elastic fluid flows through a modified boundary treatment, the Weakly Compressible Smoothed Particle Hydrodynamics technique (WC-SPH), which facilitates the efficient calculation of hydrodynamic forces in multi-body problems. In addition, consistent spatial derivative schemes were used along with a modified mass conservation equation in order to alleviate the need for artificial viscosity and/or artificial stress. Hashemi et al. (2012) and Anghileri et al. (2011) used a coupled (Finite Element Method) FE/SPH approach in order to model the 3D high-velocity impacts that involve fluid–structure interaction. Seungtaik et al. (2009) used an impulse-based boundary force technique to simulate the interaction between body–body in 3D. Both Bouscasse et al. (2013) and Ren et al. (2015) developed Weakly-Compressible SPH models for the non-linear interactions between surface waves and floating bodies. Bouscasse et al. (2013) implemented a ghost-fluid technique, which imposes no-slip conditions on the solid walls. Ren et al. (2015) defined an improved boundary treatment scheme based on "dynamic boundary particles". More recently, Liu et al. (2014) presented a SPH–RANS model for the interaction of free surface flows with moving rigid bodies. The model is validated on several test cases involving the dynamics of a solid cylinder in a free surface flow. Ren et al. (2014) represented the 2D fluid–structure interactions of surface waves and breakwaters using a coupled numerical solution SPH–DEM (Discrete-Element Method). Finally, Sun et al. (2015) developed a SPH model for fluid–structure interactions also with moving boundaries and they implemented an improved dummy particle technique for boundary treatment. The associated validation referred to 2D and 3D test cases, which involved violent hydrodynamic impacts on rigid bodies.

Another interesting CFD method for the analysis of the fluid–structure interaction problems is the finite element method (PFEM). The key feature of the PFEM is the use of a Lagrangian description to model the motion of nodes (particles) in both the fluid and the structure domains. Nodes are thus viewed as particles, which can freely move and even separate from the main analysis domain representing, for instance, the effect of water droplets (Idelsohn et al., 2004). A mesh connects the nodes defining the discretized domain where the governing equations, expressed in an integral form, are solved as in the standard FEM (Onate et al., 2004). This method preserves all the classical advantages of the Finite Element Method (FEM) for the evaluation of the integrals of the unknown functions and their derivatives, including the facilities to impose the boundary conditions and the use of symmetric Galerkin approximations, combined with the flexibility of particle methods. In particular, Onate et al. (2011) developed a solution for the equations of an incompressible continuum using PFEM allowing the use of low order elements with equal order interpolation for all the variables. The proposed approach was applied to several fluid–soil–structure interaction problems involving large motion of solid–solid interfaces and bed erosion, among other complex phenomena. Onate et al. (2014) proposed a method based on PFEM and a stabilized Lagrangian mixed velocity–pressure formulation for modelling the motion of small and large particles that are submerged in the fluid. Zhang et al. (2015) revised the classical PFEM approach of Onate et al. (2004) for rigid bodies sliding process, such as landslide problems. This approach attempts to solve the complete nonlinear dynamic governing equations in the framework of solid mechanics, via a standard finite element procedure and it was validated on a real-world landslide that occurred in Southern China. Zhu and Scott (2014) extended the OpenSees software (McKenna et al., 2000) to incorporate fluid–structure interaction handling additional pressure and pressure gradient unknowns at the element level. Gimenez and González (2015) proposed a variant formulation of classical PFEM, called PFEM-2, to solve free-surface flows with pressure gradient discontinuities, based on a continuous enriched space for pressure while keeping the advantage of the possibility to use large time-steps. Finally, it is relevant to cite the theoretical and experimental work of Xia et al. (2011a), Shu et al. (2011) and Teo et al. (2012) to formulate, validate and successively integrate in a 2D hydraulic model (Xia et al., 2011b), a simple formulation for the stability threshold of flooded vehicles.

Recently, the authors of the present paper developed a 3D SPH model for body transport in free surface flows (Amicarelli et al.,