Journal of Fluids and Structures 54 (2015) 802–822

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Prediction of energy losses in water impacts using incompressible and weakly compressible models

S. Marrone [a,b], A. Colagrossi [b], A. Di Mascio [c], D. Le Touzé [a,*]

[a] École Centrale Nantes, LHEEA Lab. (ECN/CNRS), Nantes, France
[b] CNR-INSEAN, The Italian Ship Model Basin, Rome, Italy
[c] CNR IAC, Istituto per le Applicazioni del Calcolo "Mauro Picone", Rome, Italy

ARTICLE INFO

Article history:
Received 8 August 2014
Accepted 18 January 2015
Available online 12 March 2015

Keywords:
Water impact
Energy loss
Free-surface flow
Smoothed Particle Hydrodynamics
Level-Set
Finite Volume Method

ABSTRACT

In the present work the simulation of water impacts is discussed. The investigation is mainly focused on the energy dissipation involved in liquid impacts in both the frameworks of the weakly compressible and incompressible models. A detailed analysis is performed using a weakly compressible Smoothed Particle Hydrodynamics (SPH) solver and the results are compared with the solutions computed by an incompressible mesh-based Level-Set Finite Volume Method (LS-FVM). Impacts are numerically studied using single-phase models through prototypical problems in 1D and 2D frameworks. These problems were selected for the conclusions to be of interest for, e.g., the numerical computation of the flow around plunging breaking waves. The conclusions drawn are useful not only to SPH or LS-FVM users but also for other numerical models, for which accurate results on benchmark test-cases are provided.

1. Introduction

In many engineering fields the proper modelling and simulation of violent liquid impacts is a crucial subject. Interactions during liquid impacts on structures are physically and numerically challenging phenomena because of (i) the fragmentations and reconnections of the air–water interface; (ii) the small time–space scales involved; (iii) the non-linear interactions between the fluid and the solid surface motions. In this context the role of the basic assumption made on the nature of the liquid as an incompressible or weakly compressible medium is a major concern (see, e.g., Korobkin and Pukhnachov, 1988).

The numerical simulation of these problems has been tackled by many researchers (see, e.g., Hirt and Nichols, 1981), starting from the 1980s when the volume of fluid (VOF) algorithm (Scardovelli and Zaleski, 1999) was developed. Besides VOF approaches, in the framework of mesh-based methods another free-surface capturing algorithm that became quite popular is the Level-Set (LS) method (Osher and Sethian, 1988; Sethian, 1999). More recently, the development of particle methods has widened the simulation range of free-surface flows (see, e.g., Monaghan, 1994; Kiger and Duncan, 2012; Landrini et al., 2007), because, thanks to the meshless and Lagrangian characters of such methods, it is possible to simulate liquid fragmentation in a simpler way. For this reason, in the last twenty years hundreds of articles regarding the simulation of free-surface flows through Smoothed Particle Hydrodynamics (SPH) (see, e.g., Monaghan, 2012) or Moving Particle Semi-implicit (MPS) solvers (see, e.g., Koshizuka et al., 1998; Khayyer and Gotoh, 2008) appeared in the literature. Because of the

* Corresponding author.
E-mail addresses: andrea.colagrossi@cnr.it (A. Colagrossi), david.le-touze@ec-nantes.fr (D. Le Touzé).

http://dx.doi.org/10.1016/j.jfluidstructs.2015.01.014
0889-9746/

high realism showed by those particle methods, they are also largely used in the computer graphics community (see, e.g., Müller et al., 2003; Premoze et al., 2003).

All the numerical methods cited above can hide model inconsistencies when simulating liquid fragmentation and reconnection. For instance, in simulations involving liquid collisions, the incompressible assumption implies a partial inelastic behaviour of the fluid, as demonstrated in Szymczak (1994), and leads to instantaneous mechanical energy losses. Note that this applies only for impacts where the two surfaces touch each other on a non-punctual contact surface (e.g., flat impacts). Actually, in case of punctual contact surface at impact instant, even with arbitrary small impact angle, a different incompressible theory applies (see, e.g., Dobrovol'Skaya, 1969; Wu, 2007). For non-punctual contact surface impacts, the predicted impact pressures are unbounded and this can generate ambiguities when the aim of the simulation is the analysis of structure integrity. On the other hand, the compressible assumption yields realistic predictions of the peak pressure. Nevertheless, the use of the actual speed of sound is seldom viable when considering long-time investigations of flows at low Mach number due to the implied high computational costs (too small time steps). In these situations a weakly compressible approach can be adopted, where a smaller speed of sound is used. Indeed, when dealing with liquid flows characterised by low-Mach number one can consider the solution as a superimposition of an acoustic component and a hydrodynamic one (see, e.g., Seo and Moon, 2006). The value of the sound speed used in the simulations is therefore dictated by efficiency requirements, with the aim of simulating a flow that reasonably resembles the limit behaviour of vanishing Mach number. From this point of view, an incompressible flow can be a useful reference for long-time (with respect to the impact time-scale) integration.

This work is devoted to a critical analysis of these two approaches (weakly compressible and incompressible) and to show their consequences in the results of liquid impact simulations. Indeed, even if there is an ample literature regarding the theoretical modelling of such flows (see, e.g., Lesser and Field, 1983; Korobkin and Pukhnachov, 1988; Cooker, 2002), little attention has been paid to the consistency of the numerical solutions obtained by incompressible and weakly compressible models in this context.

Besides the liquid models, other problems are related to the numerical schemes; for example, for the different interface capturing approaches (VOF–Level Set) in mesh-based solvers, it is well known that numerical diffusion and/or conservation properties can represent an issue (see, e.g., Bøckmann and Vartdal, 2014). As far as particle methods are concerned, even if mass and momenta can be conserved exactly, they can suffer from numerical dispersion and non-physical fragmentation of the fluid domain (see e.g., Le Touzé et al., 2013; Khayyer and Gotoh, 2011).

We focus our attention on the Smoothed Particle Hydrodynamics method and on a Level-Set Finite-Volume-Method (LS-FVM).

In the SPH method the liquid is treated as a weakly compressible medium while in the LS-FVM an incompressible constraint is used. Since the two solvers selected are based on completely different numerical approaches (i.e., a Lagrangian particle method and a classical Eulerian mesh-based approach), the conclusions we draw are quite general and can be of interest also for other kinds of numerical solvers.

After introducing the governing equations and the SPH and LS-FVM models, we start our analysis on the simple problem of two impinging jets in 1D which allows discussing the energy transfer behaviours of the different models. Then problems of 2D impinging jets are discussed as prototype problems useful to address energy losses for different geometrical configurations. The role of the free-surface and the different behaviour regarding energy conservation of the solvers are discussed.

2. Governing and energy conservation equations

Consider a fluid domain Ω whose boundary, $\partial\Omega$, consists only of a free surface, $\partial\Omega_F$. A compressible flow obeys the following governing equations:

$$\begin{cases} \dfrac{D\rho}{Dt} = -\rho \,\mathrm{div}\,(\boldsymbol{u}), \\ \dfrac{D\boldsymbol{u}}{Dt} = \boldsymbol{g} + \dfrac{\mathrm{div}(\mathbb{T})}{\rho}, \\ \dfrac{De}{Dt} = \dfrac{\mathbb{T}:\mathbb{D}}{\rho}, \quad p = f(\rho, e), \end{cases} \tag{2.1}$$

where D/Dt represents the Lagrangian derivative, $\boldsymbol{u}$ the flow velocity, ρ the fluid density, e the specific internal energy, $\mathbb{T}$ the stress tensor, $\mathbb{D}$ the rate of strain tensor and $\boldsymbol{g}$ can be a generic specific body force. In the present paper it represents the acceleration of gravity $\boldsymbol{g} = -g\boldsymbol{k}$, $\boldsymbol{k}$ being the upward unit vector. Thermal conductivity effects are here neglected.

If we assume the fluid to be Newtonian, the stress tensor is

$$\mathbb{T} = -p\mathbb{1} + \mathbb{V} = (-p + \lambda \,\mathrm{tr}\,\mathbb{D})\mathbb{1} + 2\mu\mathbb{D}, \tag{2.2}$$

where $\mathbb{V}$ is the viscous stress tensor, and with $\mathbb{D} = (\nabla\boldsymbol{u} + \nabla\boldsymbol{u}^T)/2$. Finally, μ and λ are the Lamé viscosity coefficients.