Available online at www.sciencedirect.com
ScienceDirect
Computers and Structures 85 (2007) 879–890
Computers & Structures
www.elsevier.com/locate/compstruc

Numerical simulation of fluid–structure interaction by SPH

Carla Antoci *, Mario Gallati, Stefano Sibilla

Dipartimento di Ingegneria Idraulica e Ambientale, Università degli Studi di Pavia, via Ferrata, 1, 27100 Pavia, Italy

Received 31 May 2006; accepted 2 January 2007
Available online 15 February 2007

Abstract

A Lagrangian model for the numerical simulation of fluid–structure interaction problems is proposed in the present paper. In the method both fluid and solid phases are described by smoothing particle hydrodynamics: fluid dynamics is studied in the inviscid approximation, while solid dynamics is simulated through an incremental hypoelastic relation. The interface condition between fluid and solid is enforced by a suitable term, obtained by an approximate SPH evaluation of a surface integral of fluid pressure.

The method is validated by comparing numerical results with laboratory experiments where an elastic plate is deformed under the effect of a rapidly varying fluid flow.

Keywords: Fluid–structure interaction; Smoothed particle hydrodynamics; Interface conditions

1. Introduction

In many engineering applications, the forces exerted by a fluid flow on the confining solid boundaries do not modify significantly the geometry of the boundaries. In this cases, the fluid flow can be studied as occurring within rigid boundaries, and the forces applied on the solid boundaries can be obtained after the characteristics of the fluid motion have been determined.

On the other hand, whenever the characteristic times of the motion of the fluid flow and of the solid boundaries are comparable, it is necessary to couple the dynamics of the two media. These fluid–structure interaction (FSI) problems can be solved by employing either a simultaneous (or direct) solution or a partitioned (or iterative) solution. A description of the two procedures can be found in [1], together with the explanation of their main advantages and drawbacks. The simultaneous technique is particularly convenient when the interaction between the structure and the fluid is very strong (and the displacements of the structure are important). Structures are usually described by Lagrangian formulations, whereas fluids are often described by Eulerian formulations. The coupling of the two media is usually obtained by an Arbitrary-Lagrangian–Eulerian (ALE) formulation for the fluid. Significant contributions [1–3] have been proposed in the simulation of FSI problems in this context. Rugonyi and Bathe [1] perform a simplified stability analysis of the interface equations and study the long-term dynamic stability of FSI systems by use of Lyapunov characteristic exponents. They also show the solution of some FSI problems, as the dynamics of spring-loaded valves in fuel pumps, that indicate the actual possibility to simulate complex coupled phenomena. Recent developments in the simulation of viscous incompressible and compressible fluid flows with structural interactions are discussed in [2]. Le Tallec and Mouro [3] simulate the dynamics of an hydroelastic shock absorber adopting an ALE formulation for the fluid equations.

An alternative approach to the numerical simulation of FSI problems consists in the description of both the fluid and the structure motion by a Lagrangian formulation. This can be especially effective when studying problems characterized by large displacements of the fluid–structure interface and by a rapidly moving fluid free-surface. An

* Corresponding author. Tel.: +39 0382 985321; fax: +39 0382 985589.
E-mail address: carla.antoci@unipv.it (C. Antoci).

0045-7949/$ - see front matter
doi:10.1016/j.compstruc.2007.01.002

example of these problems is the FSI inside safety valves for pressure reduction, where an elastic plate deforms owing to water pressure, allowing part of the fluid to flow out at atmospheric conditions, thus causing a pressure relief in the connected pipe. In this kind of problems, the use of Lagrangian techniques for both the solid and the fluid part of the problem appears promising, as it permits to easily follow in time the motion of the fluid–solid interface and to simulate the free-surface of the fluid without any specific treatment. In particular, encouraging results have been recently obtained by the smoothed particle hydrodynamics (SPH) technique (see [4] for a recent review of the method), which allows to obtain numerical solutions of the continuum equations by defining the variables at a set of suitable moving points, reconstructing the continuous field by means of interpolation functions centred on each moving point.

The SPH technique was first developed in astrophysics by Lucy [5] and by Gingold and Monaghan [6]. It was then successfully applied to the study of various fluid dynamics problems, such as free-surface incompressible flows [7], and viscous flows [8,9]. Since the early 1990s, SPH was applied also to the simulation of elasticity and fragmentation in solids: in particular, Libersky et al. [10] modelled the elastic response of solid structures by an incremental formulation of Hooke's law.

SPH has been also used to simulate the interaction between different fluids [11,12], different solids [13] and between fluids and structures [14] in presence of explosions. In some commercial codes, an SPH description of the fluid motion is coupled to a finite element formulation for the solid dynamics, in order to simulate FSI problems.

The present paper discusses a FSI model where both the fluid and the solid parts are modelled by SPH. Aim of the model is the analysis of FSI problems where large elastic displacements of the solid occur, while rapidly moving free-surfaces characterize the fluid motion.

The reliability of the numerical results yielded by the proposed SPH FSI model is checked against laboratory data obtained during a simple 2D interaction experiment.

2. Numerical model

2.1. Equations of motion

The motion of a continuum subjected to the action of gravity, in isothermal conditions, is described by the continuity equation

$$\frac{\mathrm{D}\rho}{\mathrm{D}t} + \rho\frac{\partial v_i}{\partial x_i} = 0, \tag{1}$$

and by the momentum equation

$$\rho\frac{\mathrm{D}v_i}{\mathrm{D}t} = \rho g_i + \frac{\partial \sigma_{ij}}{\partial x_j}, \tag{2}$$

where t is time, ρ is density, v_i is the velocity vector, x_i is the position vector, g_i is the gravity vector, σ_{ij} is the stress tensor and the notation implies summation over repeated indices.

The stress tensor can be decomposed into its isotropic and deviatoric parts:

$$\sigma_{ij} = -p\delta_{ij} + S_{ij}, \tag{3}$$

where $p = -\sigma_{kk}/3$ is pressure, S_{ij} is the deviatoric stress tensor and δ_{ij} is the Kronecker tensor.

Pressure can be formally defined in the same way for both fluid and solid by the following linearized equation of state, which holds for small variations of density:

$$p = c_0^2(\rho - \rho_0), \tag{4}$$

where $c_0 = \sqrt{\frac{\varepsilon}{\rho_0}}$ for the fluid and $c_0 = \sqrt{\frac{K}{\rho_0}}$ for the solid, being ε the compressibility modulus of the fluid and K the bulk modulus of the solid. Eq. (1) is strictly valid only for compressible flows, while for incompressible flows it reduces to the divergence-free condition for the velocity field. However, in order to avoid the complexity of the implicit computation of pressure in a meshless method, the incompressible fluid can be studied as weakly compressible, thus retaining the validity both of (1) and of the equation of state (4). However, since the stability of an explicit numerical integration of Eqs. (1) and (2) depends on the Courant condition, the maximum time step is inversely proportional to the sound speed c_0. It is therefore often necessary to assign to the compressibility modulus a value which is lower than the real one, in order to limit the computational time. This leads to errors that can be reduced if a proper value is assigned to ε. In particular Monaghan [7] suggests that, in order to limit density fluctuations to $\sim$1%, the Mach number, i.e. the ratio between the local flow velocity and c_0, must be everywhere lower than 0.1. Many applications of weakly compressible SPH models (see, for instance [9,11,15,16]) confirm that incompressible flows can be simulated with good precision in this way.

If the dynamics of the fluid flow is dominated by inertial forces, viscosity effects can be safely neglected, and $S_{ij} = 0$ can be assumed for fluids. For solids, the linear elastic relation between stress and deformation tensors can be derived in time in order to obtain an evolution equation for S_{ij}. The use of the corotational, or Jaumann, time derivative guarantees that the formulation is independent from superposed rigid rotations, resulting in the incremental formulation of Hooke's law corrected by the Jaumann rate:

$$\frac{\mathrm{D}S_{ij}}{\mathrm{D}t} = 2\mu\left(D_{ij} - \frac{1}{3}\delta_{ij}D_{ij}\right) + S_{ik}\Omega_{jk} + \Omega_{ik}S_{kj}, \tag{5}$$

where

$$D_{ij} = \frac{1}{2}\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\right) \tag{6}$$

is the rate of deformation tensor,

$$\Omega_{ij} = \frac{1}{2}\left(\frac{\partial v_i}{\partial x_j} - \frac{\partial v_j}{\partial x_i}\right) \tag{7}$$

is the spin tensor and μ is the shear modulus.