Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2013,25(5):673-682
DOI: 10.1016/S1001-6058(13)60412-6
www.sciencedirect.com/science/journal/10016058

Numerical simulation of hydro-elastic problems with smoothed particle hydrodynamics method*

LIU Mou-bin (刘谋斌), SHAO Jia-ru (邵家儒)
Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China, E-mail: liumoubin@imech.ac.cn
LI Hui-qi (李慧琦)
School of Information and Electronics, Beijing Institute of Technology, Beijing 100081, China

(Received December 18, 2012, Revised May 27, 2013)

Abstract: Violent free surface flows with strong fluid-solid interactions can produce a tremendous pressure load on structures, resulting in elastic and even plastic deformations. Modeling hydro-elastic problems with structure deformations and a free surface breakup is difficult by using routine numerical methods. This paper presents an improved Smoothed Particle Hydrodynamics (SPH) method for modeling hydro-elastic problems. The fluid particles are used to model the free surface flows governed by Navier-Stokes equations, and the solid particles are used to model the dynamic movement and deformation of the elastic solid objects. The improved SPH method employs a Kernel Gradient Correction (KGC) technique to improve the computational accuracy and a Fluid-Solid Interface Treatment (FSIT) algorithm with the interface fluid and solid particles being treated as the virtual particles against their counterparts and a soft repulsive force to prevent the penetration and a corrective density approximation scheme to remove the numerical oscillations. Three typical numerical examples are simulated, including a head-on collision of two rubber rings, the dam break with an elastic gate and the water impact onto a forefront elastic plate. The obtained SPH results agree well with experimental observations and numerical results from other sources.

Key words: Smoothed Particle Hydrodynamics (SPH), hydro-elasticity, Fluid-Structure Interaction (FSI), artificial stress

Introduction

Violent free surface flows with strong fluid-structure interactions are widely observed in hydrodynamics and ocean engineering. They can produce a tremendous hydro-pressure load on the solid structures and cause the structure to deform elastically or even plastically. They are usually referred to as hydro-elasticity and hydro-plasticity. For example, under extreme weather conditions, the rolling and breaking up of the water surface can produce strong slamming effects on hull structures, offshore platforms and nearby buildings, and can further lead to local damages and global instability of the structures. The large amplitude liquid sloshing in oil or Liquefied Natural Gas (LNG) ships can result in a very high impact pressure on the container, which can damage the hull walls and further lead to the leakage of oil, and even capsize ships. Therefore, how to effectively model the strong fluid-solid interaction in hydro-elasticity is very important for applications in hydrodynamics and ocean engineering.

For modeling the fluid and solid dynamics, among the grid based numerical methods, the Finite Difference Method (FDM), the Finite Volume Method (FVM) and the Finite Element Method (FEM) are most frequently used. They are currently the dominant methods in numerical simulations for solving practical problems in engineering and science. Despite the great success, the grid based numerical methods suffer from difficulties, which limit their applications in many types of complicated problems such as the hydro-elastic problems with violent deformation and even break up of the free surfaces, and movement and deformation of the solid structures. For Lagrangian grid-based methods such as FEM, a grid is attached on, moves and deforms with the moving objects. It is therefore

* Project supported by the National Natural Science Foundation of China (Grant Nos. 11172306, 11232012 and 81271650), the New Century Excellent Talents (Grant No. NCET-10-0041) and the 100 Talents Programme of the Chinese Academy of Sciences.
Biography: LIU Mou-bin (1970-), Male, Ph. D., Professor
Corresponding author: LI Hui-qi,
E-mail: huiqili@bit.edu.cn

easy to obtain the time-history of the movement and convenient to treat or track the moving features such as the free surfaces and the deformable interfaces. However it is very difficult to treat a large deformation due to possible mesh entanglement. In contrast, for Eulerian grid based methods such as FDM and FVM, a computational grid is fixed on the computational domain and there is no problem to treat large deformation. However, it is very difficult to treat or track the moving features and special algorithms are usually necessary, which are usually complicated and can induce errors.

For modeling Fluid-Structure Interaction (FSI) problems, both Eulerian and Lagrangian methods are usually used. Typical approaches include the Coupled Eulerian Lagrangian (CEL) and the Arbitrary Lagrange Eulerian (ALE). The CEL approach employs both the Eulerian and Lagrangian methods in separate (or with some overlap) regions of the problem domain. One of the most common practices is to discretize solids in a Lagrangian frame, and fluids (or materials behaving like fluids) in a Eulerian frame. The Lagrangian region and the Eulerian region continuously interact with each other through a coupling module in which the computational information is exchanged either by mapping or by special interface treatments between these two sets of grids. The ALE is closely related to the rezoning techniques for the Lagrangian mesh, and aims to move the mesh independently of the materials so that the mesh distortion can be minimized. It is in a very quick development and is widely applied to problems with large deformation and strong FSIs. Unfortunately, even with the CEL and ALE formulations a highly distorted mesh can still introduce severe errors in numerical simulations.

During the last decades, effort has been focused on the development of the next generation computational methods, the meshfree methods, such as the Moving Particle Semi-implicit (MPS)[1] method and the Smoothed Particle Hydrodynamics (SPH)[2] method. In the MPS, the governing equations are transformed into those of interactions among moving particles, and a semi-implicit algorithm is used to model the incompressible flows through solving the Poisson equation of pressure, while the other terms are explicitly calculated. The MPS method is widely applied to modeling free surface flows. The smoothed particle hydrodynamics is another popular meshfree, Lagrangian, particle method with some attractive features. The field variables (such as the density, the velocity, and the acceleration) can be obtained through discretizing the governing equations into a set of particles. The connectivity between particles is established as a part of the computation and can vary with time. Therefore, the SPH allows a straightforward handing of a very large deformation. The SPH was successfully used in solving multi-phase flows[3], heat conduction[4], elastic dynamics[5], liquid sloshing[6] and underwater explosion problems[7]. However, there are still some problems that need to be solved in the conventional SPH method, such as the stress instability, the low accuracy and the solid boundary treatment[8]. Also there are few papers dealing with the FSI problems by using the SPH method.

In this paper, an SPH model is built for simulating hydro-elastic problems with strong FSIs. The SPH model involves an improved particle approximation scheme and an enhanced fluid-solid interface treatment algorithm. In this purely meshfree model, the fluid particles are used to model the free surface flows governed by Navier-Stokes equations, and the solid particles are used to model the movement and deformation of the moving solid structures. The interface fluid and solid particles are treated by the virtual particles of their counterparts with consideration of the interaction of the neighboring fluid and solid particles as the fluid-solid interaction.

1. Equations of motion

1.1 *Governing equations*

The governing motion of the fluid flow and the solid dynamics in the isothermal condition can be described by the following continuity and momentum equations

$$\frac{\mathrm{d}\rho}{\mathrm{d}t} = -\rho \frac{\partial v_i}{\partial x_i} \tag{1}$$

$$\frac{\mathrm{d}v_i}{\mathrm{d}t} = -\frac{1}{\rho}\frac{\partial \sigma_{ij}}{\partial x_j} + f_i \tag{2}$$

where x_i, v_i, f_i, ρ denote the position, the velocity, the external force and the density, respectively. The stress tensor σ_{ij} can be decomposed into the isotropic and deviatoric parts as

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

where p is the isotropic pressure, τ_{ij} is the deviatoric viscous stress, and δ_{ij} is the Kronecker tensor.

For a Newtonian fluid such as water, the viscous shear stress is proportional to the rate of shear strain ε_{ij} ($\tau_{ij} = \mu\varepsilon_{ij}$, μ is the dynamic viscosity), and ε_{ij} can be described as

$$\varepsilon_{ij} = \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} - \frac{2}{3}\frac{\partial v_m}{\partial x_m}\delta_{ij} \tag{4}$$

Download English Version:

https://daneshyari.com/en/article/1721918

Download Persian Version:

https://daneshyari.com/article/1721918

Daneshyari.com