Available online at www.sciencedirect.com

ScienceDirect

JOURNAL OF COMPUTATIONAL AND APPLIED MATHEMATICS

Journal of Computational and Applied Mathematics 203 (2007) 376–386

www.elsevier.com/locate/cam

Modeling of two-phase flows with surface tension by finite pointset method (FPM)

Sudarshan Tiwari*, Jörg Kuhnert

Fraunhofer Institut Techno- und Wirtschaftmathematik, Gottlieb-Daimler-Strasse, Gebäude 49, D-67663 Kaiserslautern, Germany

Received 8 November 2004; received in revised form 27 June 2005

Abstract

A meshfree method for two-phase immiscible incompressible flows including surface tension is presented. The continuum surface force (CSF) model is used to include the surface tension force. The incompressible Navier–Stokes equation is considered as the mathematical model. Application of implicit projection method results in linear second-order partial differential equations for velocities and pressure. These equations are then solved by the finite pointset method (FPM), which is a meshfree and Lagrangian method. The fluid is represented as finite number of particles and the immiscible fluids are distinguished by the color of each particle. The interface is tracked automatically by advecting the color functions for each particle. Two test cases, Laplace's law and the Rayleigh–Taylor instability in 2D have been presented. The results are found to be consistent with the theoretical results.

MSC: 76D05; 76T10; 65M99

Keywords: Incompressible Navier–Stokes equations; Projection method; Two-phase flows; Meshfree method; Least-squares approximation

1. Introduction

In this paper we have presented the simulations for two-phase immiscible incompressible flows including surface tension force with variable density and viscosity. We solve the incompressible Navier–Stokes equations by applying a implicit projection method, which is based on the least-squares particle method and we call it as finite pointset method (FPM). FPM is a meshfree and fully Lagrangian particle method. The fluid domain is represented by finite number of particles (pointset), which are so-called numerical grid points and can be arbitrarily distributed. Particles move with fluid velocity and carry with them all fluid informations like density, viscosity, velocity and so on. This method is found to be appropriate for flow problems with complicated and rapidly changing geometry [13], free surface flows [22,23] and multiphase flows [9,25].

In our previous works we have presented the simulations of multiphase flows using FPM without incorporating surface tension [9,25]. The surface tension force is modeled by the continuum surface force (CSF) method [1]. The phase can be distinguished by defining the color for each fluid particle and advect the color function which results in tracking the interface accurately. The normal and the curvature of the interface can be computed from the color function. In FPM, we approximate the spatial derivatives by the weighted least-squares method. Furthermore, the application of

* Corresponding author.
E-mail addresses: tiwari@itwm.fhg.de (S. Tiwari), kuhnert@itwm.fhg.de (J. Kuhnert).

0377-0427/$ - see front matter
doi:10.1016/j.cam.2006.04.048

FPM for solving the Poisson equation has already been reported, see [10,24,25]. Several computations of flow problems by the method of least squares are handled by various authors [5,11,12,20,21,26] and references therein. Since we use the implicit projection method, we have to solve a general second-order linear partial differential equation with the Dirichlet or the Neumann boundary conditions. The scheme is second-order convergence [10,25].

A similar approach to simulate multiphase flows is the method of smoothed particle hydrodynamics (SPH). SPH was initially developed to solve the problems in astrophysics [7] and later extended to solve the several fluid dynamics problems [15,18]. The method has further been extended to simulate the multiphase flows [3,16,17]. However, the SPH has poor approximation of the second-order derivatives and is difficult to handle boundary conditions.

Since the particles move with fluid velocity, they may scatter or accumulate together. If they scatter and create some holes in the computational domain, we get some singularity in that region. So, we have to detect the holes and add new particles there. Similarly, if two particles are very close to each other, we can remove one of them in order to reduce the computational time. The proposed scheme gives accurate results compared to the theory and is tested for Laplace's law and the Rayleigh–Taylor instability.

The paper is organized as follows. In the next section we introduce the mathematical model and the numerical scheme. The ensuing section deals with FPM for solving general elliptic partial differential equations. The numerical results are presented in Section 4.

2. Mathematical model and numerical scheme

2.1. Mathematical model

We consider two immiscible fluids, for example, liquid and gas. The equations of motion of such fluids are described by the incompressible Navier–Stokes equations, which are given in the Lagrangian form

$$\frac{D\vec{v}}{Dt} = -\frac{1}{\rho}\nabla p + \frac{1}{\rho}\nabla\cdot(2\mu D) + \frac{1}{\rho}\vec{F}_{\mathrm{S}} + \vec{g}, \tag{1}$$

$$\nabla\cdot\vec{v} = 0, \tag{2}$$

where $\vec{v}$ is the fluid velocity vector, ρ is the fluid density, μ is the fluid viscosity, D is the viscous stress tensor given by $D = \frac{1}{2}(\nabla\vec{v} + \nabla^{\mathrm{T}}\vec{v})$, $\vec{g}$ is the body force acceleration vector and $\vec{F}_{\mathrm{S}}$ is the continuous surface tension force.

The surface tension force acts on the interface between the fluids. We suppose that the surface tension coefficient σ is constant. In the CSF model [1] the surface tension force per unit area $\vec{F}_{\mathrm{S}}$ is defined by

$$\vec{F}_{\mathrm{S}} = \sigma\kappa\vec{n}\delta_{\mathrm{S}}, \tag{3}$$

where $\vec{n}$ is the unit normal vector to the interface, κ is the curvature of the interface and δ_{S} is a normalized surface delta function, which is concentrated on the interface.

In this paper, we initially give a flag for each fluid particle and keep the same identification for all time. Moreover, the density and viscosity are constant on each particle path, so we have

$$\frac{\partial\rho}{\partial t} + \vec{v}\cdot\nabla\rho = 0, \tag{4}$$

$$\frac{\partial\mu}{\partial t} + \vec{v}\cdot\nabla\mu = 0. \tag{5}$$

Each fluid particle has constant ρ and μ. Since ρ and μ are discontinuous across the interface, the numerical scheme can have instabilities around such region. So, we consider the smooth density and viscosity in the vicinity of the interface. The interface region can be detected by checking the flags of particles in the neighborhood. If there are same type of flags in the neighboring list of a particle then it is far from the interface region. Near the interface particles, we have both type of flags in the neighboring list. We update the density and viscosity in each time step at each particle