Journal of Computational Physics 231 (2012) 1650–1674

Contents lists available at SciVerse ScienceDirect

Journal of Computational Physics

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

A new particle method for simulating breakup of liquid jets

Fedir V. Sirotkin, Jack J. Yoh *

School of Mechanical and Aerospace Engineering, Seoul National University, Seoul 151–742, Republic of Korea

ARTICLE INFO

Article history:
Received 18 April 2011
Received in revised form 8 September 2011
Accepted 24 October 2011
Available online 3 November 2011

Keywords:
Smoothed particle hydrodynamics
Kernels
Surface tension
Tensile instability
Jet breakup

ABSTRACT

A corrected smoothed particle hydrodynamics (CSPH) method for simulating two-phase flows including surface tension is presented. The effects of the instability in the compressional regime and particle deficiency are suppressed by adopting a new smoothing kernel and its gradient corrections. The insensitivity to the compressional instability of the adopted method is confirmed by numerical tests. The method is validated and calibrated through a series of standard numerical tests and showed quantitative agreements. The method is extended to two-dimensional jet breakup problems and provided good comparison to the theoretical and experimental results. In particular, the numerical model exhibits the transition from the jetting to dripping when the Weber number is close to its theoretical critical value.

1. Introduction

Jet breakup appears in many engineering applications that include fuel atomization in aircraft and automotive engines where the breakup influences engine efficiency significantly [1]. Nano-jet and microjet injection is an interesting research topic in the field of controlled release of drugs [2,3] where the breakup length has a direct correlation to the depth of drug penetration. Liquid jet dynamics probes a wide range of physical properties such as liquid surface tension, viscosity and density of liquid in contrast with its environment. On a small scale, typically nanometers, jets are sensitive to thermal fluctuations. On a large scale, however, gravitational interactions are important. The complexity of physical factors makes the jet phenomena quite sophisticated. In an ideal case where some of the physical factors can be neglected, the analytical study of the jet phenomena is possible.

In this study, a simulation of jet breakup released from a nozzle is analyzed using a modified smoothed particle hydrodynamics (SPH) method. While originally developed for astrophysical applications [4,5], SPH has been successfully applied to a wide range of engineering problems. The SPH is Lagrangian based method in which the fluid flow is represented by fluid pseudo-particles. These individual particles interact with one other, moving with the flow and carrying with them all of the information about the fluid. The great advantage of SPH is its flexibility and relative programming simplicity. For the detailed information on the SPH and its possible applications, we refer the reader to the review of the method [6].

The implementation of computational fluid dynamics for jet breakup problem is generally based on the Eulerian description of the fluid flow. Eulerian based modelling techniques have been heavily studied since the 1950's. Because SPH method is not as mature as well developed grid based approaches, to date there are only a few attempts of application of the particle based methods, in particular, a jet breakup problem can be found in the literature. Shibata et al. [7] applied MPS method (Moving Particle Semi-implicit Method) to study the jet breakup of inviscid liquid that is released from a nozzle and resulting droplet size distribution. Ganzenmüller et al. [8] applied SPH method for the diesel injection. In both cases the adopted range

* Corresponding author. Tel.: +82 2 880 9334; fax: +82 2 887 2662.
E-mail addresses: fyodor@snu.ac.kr (F.V. Sirotkin), jjyoh@snu.ac.kr (J.J. Yoh).

0021-9991/$ - see front matter © 2011 Elsevier Inc. All rights reserved.
doi:10.1016/j.jcp.2011.10.020

of initial liquid velocities is relatively high implying that the liquid exhibits jetting response. In this work we concentrated on the regime when the liquid exhibits jetting and dripping response.

Although SPH has a lot of advantages, it suffers from several drawbacks. As it is shown by Swegle et al. [9], SPH suffers from a tensile instability if a material experiences a high strain. This instability manifests itself as a clustering of particles, which resembles fracture and fragmentation, but is in fact a numerical artifact. The particle clustering occurs when the kernel properties and those of the physical process under consideration satisfy certain conditions. For instance, if a material experiences the compression, the instability grows where the second derivative of the kernel function is negative [9]. In order to highlight the specific cause of the instability, hereafter we will use the term compressional instability in the case when an instability grows due to the compression. Monaghan [10] proposed artificial forces proportional to the fluid pressure and the stress tensor. Dyka and Ingel [11] proposed a solution by using a non-collocated discretization of stress and velocity points. At one set of points, the stresses are evaluated while the momentum equation is calculated at another set of points. If the properties of the modeled process are known a priori, then it is possible to reduce the instability by choosing the suitable kernel. For instance, Johnson and Biessel [12] proposed a kernel function with the positive second derivative for impact problems when a material experiences a high compression.

The particle inconsistency is another numerical problem which can lead to a low accuracy and originates from the particle approximation procedure adopted in SPH. The particle inconsistency is a manifestation of the discrepancy between the spatially discretized equations and the corresponding kernel approximations in continuous form. This numerical problem has direct impact on the accuracy of the SPH approximation near the free boundaries. Various methods have been developed to solve particle inconsistency problem. In order to achieve a higher order of consistency, Liu et al. [13] introduced a corrective kernel function. The moving least square approximation was introduced into SPH computations in [14,15]. Using Taylor series expansion, Chen and Beraun [16] developed a corrected smoothed particle hydrodynamics method (CSPM). A similar idea was used to develop so-called modified finite particle methods (FPM) [17,18].

Fang et al. [18] showed that the correction of the SPH method by enforcing higher order consistency reduces effects of the tensile stability. Although FPM method is quite promising for it resolves both above mentioned numerical problems, it requires inversion of a 3×3 matrix for each particle in 2D case, which can significantly increase the computational cost. In this work, we use compromise solution which incorporates the application of the suitable kernel and kernel gradient correction [19]. Since the gradient correction requires inversion of 2×2 matrix in 2D case for each particle, the computational cost of our approach is smaller than that for FPM.

Several attempts have been made to model the effect of surface tension within the framework of SPH. To date two general approaches can be distinguished. The first one is the application of Continuum Surface Force (CSF) method [20], where the surface tension is modeled as a volume force in a narrow region close to the interface. Morris [21] describes a method where the surface tension is modeled using the curvature of the interface and surface delta function which are computed by means of a color field. This model has several drawbacks. Calculating the curvature is error-prone close to the surface where particles have only a few neighbors. Further, calculating the second derivative is sensitive to particle disorder. Hu and Adams [22] solve the above-mentioned issues of [21]. They express the surface tension as the divergence of the stress tensor which is defined by the color field. In such a way they exclude an explicit computations of curvature. The second approach models the surface tension as cohesive forces among the particles, which cancel out in the bulk of the fluid [23,24]. This mimics the physical nature of the surface tension. In this work we applied improved version of CSF method [21].

The paper is organized as follows: In Section 2, the governing equations of SPH and corrected SPH methods are outlined. In addition, issues connected with the interface tracking and kernel's choice are considered. In Section 3, numerical tests of stretching drop, Poiseuille flow and oscillating rod are presented to demonstrate the validity and ability of the method to simulate viscous fluid flows. The effect of the tensile instability is also considered, and a calibration of the numerical parameters required for the subsequent section is performed. The present method is applied to simulation of the jet breakup problem in Section 4. The concluding remarks are reported in Section 5.

2. Numerical method

2.1. Smoothed particle hydrodynamics

Kernel interpolants are the basis of SPH method. Any function $f(\mathbf{x})$ can be defined through its convolution with a kernel function, W, as

$$f(\mathbf{x}) = \int f(\mathbf{x})W(\mathbf{x} - \mathbf{x}', h)d\mathbf{x}', \tag{1}$$

where h is the smoothing length defining the influence domain of the kernel function. The latter has two properties

$$\int W(\mathbf{x} - \mathbf{x}', h)d\mathbf{x}' = 1 \tag{2}$$

and

$$\lim_{h \to 0} W(\mathbf{x} - \mathbf{x}', h) = \delta(\mathbf{x} - \mathbf{x}'), \tag{3}$$

where $\delta(\mathbf{x} - \mathbf{x}')$ is Delta function.