INTERNATIONAL JOURNAL OF
NUMERICAL ANALYSIS AND MODELING
Volume 12, Number 2, Pages 268–285

PHASE FIELD SIMULATION OF DROP FORMATION IN A COFLOWING FLUID

JIEWEI LIU* AND XIAO-PING WANG

Abstract. We numerically investigate the dynamics of drop formation when a Newtonian fluid is injected through a tube into another immiscible, co-flowing Newtonian fluid with different density and viscosity using the phase field method. The two phase system is modeled by a coupled three dimensional Cahn-Hilliard and Navier-Stokes equation in cylindrical coordinates. And the contribution from the chemical potential has been taken into account in the classical Navier-Stokes equation. The numerical method involves a convex splitting scheme for the Cahn-Hilliard equation and a projection type scheme for the momentum equation. Our study of the dynamics of the drop formation is motivated by the experimental work by Utada *et al* [*Phys. Rev. Lett.* **99**(2007), 094502] on dripping and jetting transition. The simulation results demonstrate that the process of drop formation can be reasonably predicated by the phase field model we used. Our simulations also identify two classes of dripping to jetting transition, one controlled by the Capillary number of the outer fluid and another one controlled by the Weber number of the inner fluid. The results match well with the experimental results in Utada *et al* [A. S. Utada, A. Fernandez-Nieves, H. A. Stone, and D. A. Weitz, *Phys. Rev. Lett.* **99**(2007), 094502] and Zhang [*Chem. Eng. Sci.* **54**(1999), 1759-1774]. We also study how the dynamics of the drop formation depends on the various physical parameters of the system. Similar behaviors with existing results are obtained for most parameters, yet different behavior is observed for density ratio λ_ρ and viscosity ratio λ_η.

Key words. Two phase flows, coflowing, phase field method, dripping, jetting.

1. Introduction

Dispersion of one fluid into another fluid through a vertical tube is of great importance in scientific research because of its widespread applications in the industrial production, like copolymers, cosmetics, capsules and pharmaceutics (Hua et al. [16]). The potential use of the dispersion technology is always limited by its ability to precisely control the size distribution of the droplets (Carlson et al.[5]). Over the last decade, many experiments have been carried out (A. M. Gañán-Calvo [9], Umbanhowar et al. [31], Gañán-Calvo et al. [10], Cramer et al. [6], Garstecki et al. [11], Utada et al. [30]), aiming at developing technologies to produce mono-disperse droplets with controllable size. It is found that a coflowing outer fluid or flow-focusing technique could produce smaller drops and give rise to mono-dispersion (Chuang et al. [7], Gañán-Calvo et al. [10], Utada et al. [30]).

Numerical simulation serves as a good complementary to the experimental investigation and theoretical analysis. Numerical methods for simulating multi-phase problems can be divided into two classes: sharp interface method and diffuse interface method. The advantage of the diffuse interface method is its ability to handle

Received by the editors March 9, 2014.
2000 *Mathematics Subject Classification.* 65M12, 65M70, 65P99.
*Corresponding author.
This publication was based on work supported in part by Award No SA-C0040/UK-C0016, made by King Abdullah University of Science and Technology (KAUST), Hong Kong RGC-GRF grants 603107, 604209 and 605311. This work forms part of the first author's PhD thesis under the supervision of the second author. The first author thanks the HKUST for providing Postgraduate Studentship.

the topological change of the interface which is important in the current application to the drop formation and dynamics.

Many numerical studies have been carried out for the coflowing fluid-fluid system using the sharp interface methods. Oguz and Prosperetti [19] studied the dynamics of gas bubble growth and detachment in a liquid for ir-rotational flow using a boundary integral method. As a complementary to the research by Oguz and Prosperetti [19] that focused mainly on inertial effect, Wong et al. [32] studied the motion of a pinned gas bubble expanding or contracting from a submerged capillary tip for flows with low Reynolds number using the boundary integral method. Zhang and Stone [36] studied drop formation in a quiescent and coflowing fluid by solving the governing Stokes equation using the boundary integral method, with the main focus on the assessment of the influence of three dimensionless number on drop evolution and breakup. In a series papers, Richards et al. [22, 23, 24] developed a robust and stable numerical method which combined the volume-of-fluid (VOF) method [15] and the continuous-surface-force (CSF) method [2] to simulate liquid-liquid systems. Using the same numerical method as Richards et al. [22, 23, 24], Zhang [34] investigated the drop formation dynamics in the dripping region and found good agreement with his experiment. More recently, Suryo and Basaran [27] studied the tip streaming forming from a tube in a coflowing outer fluid under creeping flow conditions. They solved the Stokes equations using the Galerkin finite element method for spatial discretization and adaptive finite difference method for time integration.

Diffuse interface method has also been used to simulate the drop formation and dynamics. Zhou et al. [35] investigated drop formation in the quiescent air and flows in a flow-focus device. The dynamics of drop formation can be classified into two regimes. One is dripping, and the other is jetting. Previous research on coflowing fluid mainly focus on the dynamics of liquid drop or gas bubble. The transition from dripping to jetting has not been studied numerically, to the author's knowledge. In this paper, we give a systematic numerically studies of drop formation dynamics in a three dimensional coflowing fluid-fluid system in cylindrical coordinates. The motion of the interface is modeled by a diffuse interface model consisting of the Cahn-Hilliard Navier-Stokes equations. The numerical method involves a convex splitting scheme for the Cahn-Hilliard equation and a projection type scheme for the Navier-Stokes equation. We study how the dynamics of the drop formation depends on the various physical parameters of the system. In particular, we are interested in the dripping to jetting transition behavior.

The rest of this paper is organized as follows: In section 2, we describe the mathematical formulation of the problem, including governing equations, boundary and initial conditions, in both dimensional and dimensionless form. In section 3, we present the numerical method for solving the Cahn-Hilliard Navier-Stokes equations with different density and viscosity ratio. Section 4 shows our numerical results and the comparison with the experiments. Two different classes of dripping-to-jetting transition observed in the experimental paper [29] are identified. Section 5 is the conclusion.

2. Problem formulation

In our problem, an incompressible Newtonian fluid with density ρ_i and viscosity η_i is injected through a vertical capillary tube of radius R_i into a coflowing, immiscible, incompressible Newtonian fluid with density ρ_o and viscosity η_o, the outer fluid is contained in a coaxial cylindrical tube of radius R_o. The dispersed phase