Thermal Science and Engineering Progress 8 (2018) 385–396

Contents lists available at ScienceDirect

Thermal Science and Engineering Progress

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

Heat transfer enhancement due to internal circulation within a rising fluid drop

T.V. Binu, S. Jayanti*

Department of Chemical Engineering, Indian Institute of Technology Madras, Chennai 600036, India

ARTICLE INFO

Keywords:
Heat transfer
Rising drop
Internal circulation
Convection
Diffusion
Computational fluid dynamics

ABSTRACT

Motion of drops and bubbles plays a critical role in the efficiency of industrial operations such as extraction. As the drop moves in the continuous phase, the tangential shear stress induced by the continuous fluid at the drop interface results in internal circulation, which may enhance transport of components and energy between the dispersed phase (drop) phase and the continuous phase. In the present study, computational fluid dynamics (CFD) simulations have been used to study this aspect in detail. Coupled calculations for the two phases have been carried out for fluid flow over a spherical, non-deformable drop of a given diameter. The predicted internal circulation is validated by comparing with analytical results for very low Reynolds numbers and experimental data at higher Re. The calculation framework is then used to study the convective-diffusive heat transfer to the drop for various combinations of Re and Pr. The heat transfer within the drop is found to comprise of three regimes, two diffusion controlled heat transfer regimes (initial and final) connected by an intermediate convection controlled heat transfer regime. The heat transfer enhancement by convection has been deduced and is found to depend both on Re and Pr. A dimensionless correlation has been developed to predict the convective heat transfer enhancement.

1. Introduction

Dispersions of bubbles and drops in liquids are often encountered in industrial operations. In processes such as liquid-liquid extraction, the dispersion of one fluid in another fluid and the motion of bubbles and drops play a critical role in the efficiency of operation. Unlike rigid spheres, the tangential stress caused by the continuous fluid at the interface results in internal circulation within the drops which reduces the drag and hence a fluid sphere moving in an immiscible liquid will have a higher velocity compared to a solid sphere of same size and density [1–3]. In the presence of impurities, the interfacial motion will be hindered and the fluid sphere will behave more or less like a solid sphere [4,5]. The rate of heat transfer and mass transfer of solute between the outer fluid and the drop can be affected by this internal circulation [6,7] and hence the hydrodynamics study of motion of drops through an immiscible fluid is important. For example, the Flue Gas Desulfurization (FGD) process used to remove SO_2 present in flue gas from a coal or oil fired power plant involves contacting of spray of lime or limestone slurry with the flue gas [8–10]. The relative motion between the slurry drops and flue gas causes circulation within slurry droplets which in turn influences the absorption of SO_2 at the interface and subsequent dissolution and reactions occurring inside the drop and the heat transfer associated with these reactive processes. Analytical expressions to predict hydrodynamics are available only for low Reynolds numbers [11] and for higher Reynolds numbers correlations based on experiments are to be used for the prediction. But the experimental results from different set ups can be different as the degree of contamination varies from one set-up to another [3]. Moreover, in some cases, like FGD and pollutant absorption in clouds, the Reynolds number falls in the intermediate range, where correlations and experiments are limited.

Hadamard and Rybczynski [1,11] solved the equations of motion for circulating drops in the creeping flow regime and obtained an expression for terminal velocity, which is given by

$$U = \frac{2}{3}\frac{(1+\lambda)}{(2+3\lambda)}\frac{R^2 g(\rho_o - \rho_i)}{\mu_o}$$

where U is terminal velocity, λ is ratio of viscosities, $\frac{\mu_i}{\mu_o}$, R is radius of drop, ρ is density and the subscripts 'i' and 'o' represent internal (drop side) and external (continuous phase) respectively.

Garner [2] studied circulation inside rising gas bubbles by visual and indirect means. The steady motion of a liquid drop in another liquid of comparable density and viscosity was studied and a correlation was

* Corresponding author.
E-mail address: sjayanti@iitm.ac.in (S. Jayanti).

https://doi.org/10.1016/j.tsep.2018.09.009
Received 1 June 2018; Received in revised form 25 September 2018; Accepted 25 September 2018

developed by Harper and Moore [12] for high Re. Le Clair et al. [13] were the first ones to use finite difference method to study steady motion of water drops falling at terminal velocity in air. Abdel-Alim and Hamielec [14] and Rivkind and Ryskin [15] developed predictive equations to estimate drag coefficient for liquid- liquid systems for low to moderate Re. Oliver and Chung [16] pointed out that there was a discrepancy of 20% between these two equations and they failed to match with known results for the limiting cases of Reynolds number, namely, the case of Hadamard-Rybczynski [1,11] at very low Re and the case of Harper and Moore [12] at high Re. They proposed a theoretical prediction of drag coefficient for Re between 0 and 2. Bhaga and Weber [17] determined shapes and terminal rise velocities of bubble rising in viscous liquids experimentally and visualized flow field around a rising bubble through hydrogen bubble tracer technique. Shape regimes and terminal rise velocities were correlated for Morton number greater than 0.004 as a function of Re alone. Buoyancy-driven motion of a deformable drop through a quiescent liquid was studied by Dandy and Leal [18] for a wide range of Reynolds numbers ($0.005 \leq Re \leq 250$), Weber numbers ($0.005 \leq We \leq 14$), viscosity ratios ($0.001 \leq \lambda \leq 1000$) and density ratios ($0.001 \leq \zeta \leq 1000$), numerically. Wegener et al. [3] measured the rise velocity of single toluene droplets rising in water ranging from 1 mm to 7 mm, experimentally. They observed that for small droplets (diameter < 2.2 mm), no significant drop deformation occurred, for 2.2 mm drops fluctuations in velocity occurred initially, for drop diameter 2.4 mm velocity reduced rapidly and oscillated around a lower value, and 3 mm drops were oblate but drop shape was preserved with maximum and characteristic mean velocity close to each other. Drop rise velocities for three systems representing high, intermediate and low interfacial tension were investigated experimentally and numerically using the finite element modeling (FEM) code NAVIER by Baulmer et al. [5].

There are fewer studies of heat and mass transfer within drops with internal circulation. Elperin et al. [19] observed a rise in temperature at the gas-liquid interface due to heat released during absorption by a falling liquid droplet. This increase in temperature reduces the equilibrium absorbate concentration resulting in a decrease in the driving force for mass transfer and mass flux during absorption. Wylock et al. [20] studied numerically the mass transfer of a component in gas phase into a liquid droplet in free fall followed by reaction of the absorbed component in the liquid phase. Mass transfer in droplets moving in a quiescent ambient liquid was studied numerically by Wegener et al. [21] without considering chemical reaction, heat transfer and surfactants. The need to consider mutual coupling of fluid dynamics and mass transfer in the case of droplets was discussed by them.

The objective of the present work is to study the heat transfer within a fluid drop moving through another immiscible fluid which is at a higher temperature. The hydrodynamics for this was obtained by modeling the two-phase flow system as two single phase flow systems coupled through shear and velocity boundary conditions in a computational fluid dynamics (CFD) simulation platform. This enables accurate treatment of the interface in terms of the momentum and mass exchanges across it. It also obviates the need for a time-dependent solution which is required when conventional two-fluid models are used in combination with the volume of fluid (VOF) method [22] or the level-set (LS) method [23] for the resolution of the interface. The need for very fine grids, the attendant very small time steps required for a stable solution, and the need for interface reconstruction make these methods computationally expensive [24]. These problems can be avoided in the present approach.

2. Problem formulation

2.1. Hydrodynamics

The two-phase flow problem of movement of a spherical drop in an immiscible liquid is decoupled into two single phase flow problems,

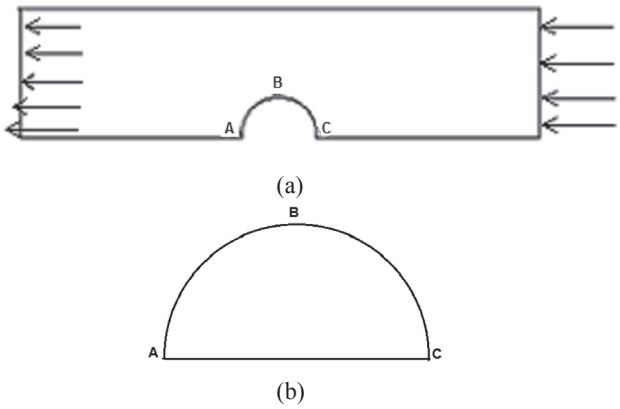

Fig. 1. Problem formulation of flow over and inside a rising drop: (a) computational model (Model 1) for external flow calculation, and (b) computational model (Model 2) for the calculation of flow inside the drop.

namely, Model 1 and Model 2, representing respectively the flow over and flow inside the drop. Here, Model 1 constitutes the continuous phase (Fig. 1a) and Model 2 represents the drop/bubble (Fig. 1b). In both models, single-phase, constant-property, laminar form of Navier-Stokes equations are solved using an unstructured grid formulation. A series of simulations, alternating between Model 1 and Model 2, are done in which the results from Model 1 are used to specify the interface boundary conditions for Model 2, and vice versa. Model 1 has a specified uniform inlet velocity on the right, a fully developed flow boundary condition at the outlet on the left side and symmetry planes at the bottom and the top except for the curved part ABC representing the drop over which spatially-varying shear stress, obtained from Model 2, is specified. Model 2 consists of one half of the cross-sectional plane of the axi-symmetric drop. It has a symmetry plane as a boundary condition along the bottom boundary and a specified velocity, obtained from Model 1, as the boundary condition along the curved boundary. In both domains, single phase Newtonian fluid flow prevails. The two fluids have different densities and viscosities which are assumed to be constant throughout each domain. The values of these variables have been used to arrive at target viscosity ratios and Reynolds numbers.

The computations have been done using the CFD simulation tool ANSYS FLUENT. A spherical, non-deformable drop of 1 mm was considered for simulation and both continuous and dispersed phases were assumed to be incompressible. Taking advantage of the rotational symmetry of both flows, steady, two-dimensional, axisymmetric simulations were performed to resolve the flow field in the two domains. To start with, Model 2 was considered with a moving wall boundary condition for the interface (shown as ABC in Fig. 1) and an initial translational velocity was assumed (estimated from the analytical model of Hadamard and Rybczynski [11]) for iteration 1 of Model 2. The shear obtained at the interface for this calculation of Model 2 was used as boundary condition in the first iteration for Model 1. The rise velocity of the drop was calculated for each Reynolds number using Hadamard-Rybczynski expression and was used as inlet velocity for Model 1. This was done to keep the drop stationary, with the continuous fluid flowing past it. The velocity at the wall from Model 1 was fed back to Model 2 as a boundary condition for the interface. In subsequent calculations of Model 1, the inlet velocity, which effectively becomes the rise velocity of the drop, was varied so as to ensure that the computed (from wall shear and surface pressure) remained constant and equal to the net buoyancy force acting on one-half of the drop. This successive set of calculations was continued till convergence.

At the end of the iterative calculation, one would have matched shear stress and velocity boundary conditions across the surface ABC in both Model 1 and Model 2 for specified fluid properties, drop size and iteratively-determined inlet velocity of Model 1 which is also the rise