International Journal of Heat and Mass Transfer 97 (2016) 653–661

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Modeling and simulation of liquid–liquid droplet heating in a laminar boundary layer

E.A. Wenzel, F.A. Kulacki, S.C. Garrick *

Department of Mechanical Engineering, 111 Church Street SE, University of Minnesota, Minneapolis, MN 55455-0111, USA

ARTICLE INFO

Article history:
Received 1 July 2015
Received in revised form 19 February 2016
Accepted 21 February 2016

Keywords:
Droplet heating
Multiphase
Magnus force

ABSTRACT

Asymmetric liquid–liquid droplet heating mechanisms differ from the more commonly studied and better understood symmetric liquid–gas mechanisms. In this work, we simulate two-dimensional low Weber number droplet heating in developing low Reynolds number liquid boundary layers. Of particular interest are the influences of Weber, Prandtl, and Reynolds number magnitudes on the system evolution. We perform simulations with a coupled Eulerian–Lagrangian interface capturing methodology – the Lagrangian volume of fluid – alongside an Eulerian solver for the Navier–Stokes equations that provides the spatial and temporal evolution of the temperature and velocity fields for the droplet and the surrounding fluid. Our results show droplet rolling induced by the velocity boundary layer modifies the temperature field in and around the droplet. Conduction negates the thermal influence of rolling in low Prandtl number droplets, but modifies the continuous phase temperature field. The Magnus force separates the droplets from the heated surface, decreasing their heating rate. These results establish the fundamentals of asymmetric liquid–liquid droplet heating in developing boundary layers: it is necessary to include the Magnus force in physically representative near-wall droplet heating models, and resolution of near-droplet temperature gradients may be necessary in situations with temperature dependent interface processes.

1. Introduction

The mechanisms of liquid dispersed phase heating in developing liquid boundary layers as a function of phase-dependent properties are largely unknown. One impacted field is the boiling of dilute emulsions, which have been proposed for use in next-generation electronic cooling devices [25,26]. It is impossible to make high accuracy predictions of liquid–liquid system behavior without an understanding of the interactions between droplets and liquid boundary layers, particularly when attempting to predict complicated phenomena such as chain boiling [4]. The goal of the present investigation is to uncover the dynamics of droplets in the developing thermal and hydrodynamic boundary layers of a liquid continuous phase.

Numerous experimental and computational studies have been performed with the intent of understanding the hydrodynamics of droplets suspended in liquid shear flows. Droplet deformation, breakup, and coalescence in simple shear flows have been readily demonstrated between parallel plates [6,8,13,14,29,31,32,34]. Researchers have also considered droplet deformation and the Magnus force in shear flows governed by Stoke's equations [12,20,21]. Counter-rotating vortices have been physically observed inside droplets in a variety of configurations [35,36], and researchers investigating the temperature distributions within droplets have cited these vortices as a droplet heat transfer mechanism [7,10]. Behaviors revealed by these spatially resolved studies, including internal circulation and the isothermal nature of highly diffusive droplets, have lead to the development of droplet heat transfer models [28]. These models can be partitioned into five categories (six including full Navier–Stokes solutions) [27]: spatially and temporally isothermal droplets; thermally lumped droplets; solid droplets (the conduction limit); conduction limit models with an effective thermal conductivity to account for the thermal effects of convection [2,15]; and droplets with prescribed internal convection [11,22,24]. These droplet heat transfer models make significant assumptions about droplet conduction, deformation, and potential internal convection, which generally apply to liquid droplets in a gaseous continuous phase; the influence of strong surface tension coupled with the asymmetry of a developing liquid boundary layer may have unforeseen affects on the developing temperature field that are not well-satisfied by current droplet heat transfer models, rendering their use unjustified in the present context.

* Corresponding author. Tel.: +1 612 624 5741; fax: +1 612 624 5230.
E-mail address: sgarrick@umn.edu (S.C. Garrick).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2016.02.067
0017-9310/

In this study, we examine the dynamics of a droplet near a heated surface. We employ a coupled Eulerian–Lagrangian solver to model two-dimensional droplets in the developing momentum and thermal boundary layers of a liquid continuous phase in the laminar regime. We consider the effects of dispersed and continuous phase Prandtl numbers, Reynolds number, and Weber number on the droplet dynamics and heat transfer.

2. Formulation

2.1. Fluid transport

A conservative form of the Navier–Stokes equations govern multiphase flows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \tag{1}$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \mathbf{u} \cdot \nabla (\rho \mathbf{u}) = -\nabla P + \nabla \cdot \tau + \mathbf{F}_{st}, \tag{2}$$

$$\frac{\partial (\rho C_v T)}{\partial t} + \nabla \cdot (\rho C_v T \mathbf{u}) = \nabla \cdot (k \nabla T), \tag{3}$$

which are a function of the fluid density, ρ, the velocity vector, $\mathbf{u}$, the static pressure, P, the deviatoric stress tensor for Newtonian fluids, τ, the specific heat, C_v, the thermal conductivity, k, and the fluid temperature, T. The surface tension term,

$$\mathbf{F}_{st} = \sigma \kappa \hat{n}, \tag{4}$$

scales linearly with the interfacial curvature, κ, and the interfacial tension coefficient, σ, and acts along the outward unit normal, $\hat{n}$. Surface tension arises from an imbalance in intermolecular attraction and physically manifests as a surface force from a macroscopic perspective. To rectify the difficulty this poses for Eulerian numerical methods, [3] suggest the implementation of a continuum surface force model (CSF), wherein a surface tension volume force acts in a small neighborhood of the interface in a way that replicates the large scale behavior of surface tension. Appropriate selection of the neighborhood and modeling of the surface tension volume force allows for accurate numerical representation of surface tension in an Eulerian framework [23]. An additional benefit of the CSF is that it eliminates the need for fluid property jump conditions at the interface because the finite interface thickness allows for smooth variations in properties.

This study considers a liquid–liquid system but solves the compressible Navier–Stokes equations. A Cole-stiffened equation of state is implemented for pseudo-incompressibility [30],

$$P = \rho_o \frac{c_s^2}{\gamma} \left[\left(\frac{\rho}{\rho_o} \right)^{\gamma} - 1 \right], \tag{5}$$

where c_s is the speed of sound, ρ_o is a reference density, and γ is the adiabatic index. The compressible Navier–Stokes equations, coupled with the Cole-stiffened equation of state, emulate an incompressible flow system.

2.2. Multiphase representation

A Lagrangian volume of fluid (LVOF) approach [17] provides interface capturing and the surface tension force. The domain is seeded with N particles of volume V_i, each carrying a volume of fluid (VOF), ψ_i. Particles carrying a VOF of $\psi_i = 0$ belong to the continuous phase, while particles carrying a VOF of $\psi_i = 1$ belong to the dispersed phase. A weight function, W, operates on the LVOF field to construct the color function, a kernel representation of the LVOF, given by

$$c_i = \frac{\sum_{\substack{j=1 \\ j \neq i}}^{N} \psi_j V_j W(\mathbf{x}_j - \mathbf{x}_i)}{\sum_{\substack{j=1 \\ j \neq i}}^{N} V_j W(\mathbf{x}_j - \mathbf{x}_i)}. \tag{6}$$

Unit interface normal vectors, $\hat{n}_i$, are computed on each particle via

$$\hat{n}_i = \frac{\sum_{\substack{j=1 \\ j \neq i}}^{N} (c_j - c_i) V_j \nabla W(\mathbf{x}_j - \mathbf{x}_i)}{\left| \sum_{\substack{j=1 \\ j \neq i}}^{N} (c_j - c_i) V_j \nabla W(\mathbf{x}_j - \mathbf{x}_i) \right|}. \tag{7}$$

The curvature, κ_i, is then computed from the unit normal vectors according to

$$\kappa_i = -\sum_{\substack{j=1 \\ j \neq i}}^{N} \nabla W(|\mathbf{x}_j - \mathbf{x}_i|) \cdot (\hat{n}_j - \hat{n}_i) V_j. \tag{8}$$

An approach based on smoothed particle hydrodynamics and the CSF Model is used to construct the surface tension term for each Lagrangian particle [23],

$$\mathbf{F}_i = \sigma \kappa_i \mathbf{n}_i. \tag{9}$$

The Eulerian surface tension present in Eq. (2), $\mathbf{F}_{st}$, is obtained for every Eulerian cell by averaging the particle surface tension forces within each cell,

$$\mathbf{F}_{st} = \frac{1}{N_{pc}} \sum_{i=1}^{N_{pc}} k_1 \mathbf{F}_i, \tag{10}$$

where N_{pc} is the number of particles in the Eulerian cell and k_1 is a calibration constant.

3. Results

3.1. Flow description and physical parameters

We simulate a droplet of radius R_o traveling above a heated surface near a laminar boundary layer. The boundary layer is treated in a temporal manner which significantly reduces the computational time [1,9,18,19]. The relevant dimensionless numbers for this configuration are the Reynolds number, the Weber number, and the Prandtl number. The Reynolds number, $Re = \rho_c U_o L_o / \mu_c$, is based on the free-stream velocity, U_o, the distance from the surface to the bottom of the droplet, $L_o = R_o$, the continuous phase density, ρ_c, and the continuous phase viscosity, μ_c. The Weber number, $We = \rho_d U_o^2 R_o / \sigma$, is based on the interfacial tension coefficient, σ, the droplet radius, R_o, the dispersed phase density, ρ_d, and the free-stream velocity. The Prandtl number, $Pr = k/\nu$, depends on the thermal conductivity, k, and kinematic viscosity, ν, of the local phase. A schematic of the domain appears in Fig. 1.

To model a small droplet in the near-wall region, we select the Weber number to be of order 0.1 and the Reynolds number to be of order 1. This combination allows for laminar, diffusion-dominated, two-dimensional flows. These parameters additionally allow the droplets to remain relatively undeformed – a characteristic of small liquid droplets. Simulations are performed with Reynolds numbers of $Re = 1$ and $Re = 5$. For each Reynolds number, the dispersed phase Prandtl number assumes values of $Pr_d = 0.2, Pr_d = 1$, and $Pr_d = 5$, while the continuous phase Prandtl number is $Pr_c = 1$. A constant heat flux is imposed at the lower surface. Table 1 provides a summary of all simulations performed.

3.2. Numerical specifications

The Navier–Stokes equations are solved via the second order accurate MacCormack scheme. The domain size is $12L_o \times 9L_o$ and