International Journal of Heat and Mass Transfer 126 (2018) 649–659

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Asymmetric heat transfer characteristics of a double droplet impact on a moving liquid film

Yi-Bo Wang [a,b,c], Xiao-Dong Wang [a,b,c,*], Tian-Hu Wang [a,b,c], Wei-Mon Yan [d,e,**]

[a] State Key Laboratory of Alternate Electrical Power System with Renewable Energy Sources, North China Electric Power University, Beijing 102206, China
[b] Research Center of Engineering Thermophysics, North China Electric Power University, Beijing 102206, China
[c] Key Laboratory of Condition Monitoring and Control for Power Plant Equipment of Ministry of Education, North China Electric Power University, Beijing 102206, China
[d] Department of Energy and Refrigerating Air-Conditioning Engineering, National Taipei University of Technology, Taipei 10608, Taiwan
[e] Research Center of Energy Conservation for New Generation of Residential, Commercial, and Industrial Sectors, National Taipei University of Technology, Taipei 10608, Taiwan

ARTICLE INFO

Article history:
Received 27 February 2018
Received in revised form 30 May 2018
Accepted 31 May 2018

Keywords:
Double droplet
Impact
Moving liquid film
Heat transfer
CLSVOF

ABSTRACT

In this work, a coupled level-set and volume of fluid method is employed to investigate a two-dimensional double droplet impact on a moving liquid film. The flow and heat transfer characteristics are analyzed for various film moving velocities, horizontal spacings between the two droplets, impact velocities, and film thicknesses. The results show that four small jets occur near the neck at the contact regions between the droplet and film at the initial impact stage. Subsequently, the two peripheral jets expand outward and form a crown, while two inner jets merge into a central uprising jet. The largest local wall heat flux appears in the two impact regions; however, three extra peaks are observed in the jet regions due to the enhanced convection heat transfer. As compared with impact on the stationary film, the peripheral and central jets for impact on the moving film exhibit an asymmetric feature due to shearing effect of the film, leading to an asymmetric and enhanced local wall heat flux. The asymmetry becomes more significant for larger film moving velocities. The heat transfer enhancement in the impact region is more remarkable for a smaller spacing between the two droplets, a higher impact velocity, or a thinner film thickness based on the present simulation conditions.

1. Introduction

The impact of droplets on liquid films is critically important in many industrial processes, such as spray cooling, ink-jet printing, fuel injection and atomization, and so forth. Because of high heat and mass transfer capabilities, it has attracted extensive attention in recent decades [1,2]. In spray cooling, keeping stability of liquid films on hot surfaces is very crucial to maintaining its high heat transfer efficiency [3]. In thermal spraying, the formation of uniform liquid films has a positive effect on improving coating performance and reducing spraying material [4]. Icing may appear on aircraft wings during the flight when super-cooled droplets within the cloud impact the wing surfaces, leading to significant changes in the original aerodynamic configuration and hence seriously reducing the flight safety [5].

In the applications mentioned above, liquid films are commonly not stationary but moving on solid surfaces. However, most of the previous studies focused on the impact of droplets on stationary liquid films. Okawa et al. [6] experimentally studied the droplet impact on a stationary liquid film. Their results showed that the limits of splash and the amount of secondary droplets can be characterized by a dimensionless number $K = \mathrm{WeOh}^{-0.4}$. Amon et al. [7] studied the influence of surface textures on the spray cooling device through experiments. They reported that the morphological evolution of liquid film is closely associated with its cooling efficiency. Because the impact of droplets on liquid films occurs in a very short time, numerical modeling and simulations have been extensively employed to investigate this subject. Zhang et al. [8] numerically studied a single micro-droplet impact on a dry or a wet wall via a volume of fluid (VOF) method. They proposed a critical parameter K to determine the droplet splash/non-splash boundary and found that the splash took place more easily on the dry wall than on the wet one. Liang et al. [9] employed a coupled level-set and volume of fluid (CLSVOF) method to study the

---

* Corresponding author at: State Key Laboratory of Alternate Electrical Power System with Renewable Energy Sources, North China Electric Power University, Beijing 102206, China.

** Corresponding author at: Department of Energy and Refrigerating Air-Conditioning Engineering, National Taipei University of Technology, Taipei 10608, Taiwan.

E-mail addresses: wangxd99@gmail.com (X.-D. Wang), wmyan@ntut.edu.tw (W.-M. Yan).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.05.161
0017-9310/© 2018 Elsevier Ltd. All rights reserved.

impact of a single droplet on a flat liquid film. Their results demonstrated that the average wall heat flux increases with the increase in impact velocity, while the film thickness and droplet diameter have little effect. Hong and Wang [10] simulated a double droplet impact on a spherical liquid film via a CLSVOF method. They presented that the wall structure has an important influence on heat transfer characteristics and the average wall heat flux increases with the increase in spherical curvature. Wang et al. [11] numerically studied a single droplet impact on a hot liquid film under vibration environment through a VOF method. Their results indicated that the wall vibration benefits the formation of a stabilized and intact film during the impact and hence has invigorating effect on heat transfer. Jiang et al. [12] employed a VOF method to simulate the impact of a single droplet on a solid wall with a pre-existing thin film of the same liquid. They found that the splashing behavior depends on the impact velocity and fluid properties, and the dissipated heat can be enhanced by increasing the impact velocity, droplet diameter, film depth, cooling droplet, and wall temperature. Kuhlman et al. [13] also employed a VOF method to investigate a single droplet impact on a stationary liquid film. They proposed a method for calculating the local and average heat flux by the sub-cavity liquid volume and cavity lifetime.

It can be anticipated that when droplets impact a moving liquid film, dynamic behaviors of the impact inevitably become more complex and exhibit an asymmetric feature because of the presence of the shear stress between the droplets and film. However, the effects of the shear stress on the flow and heat transfer characteristics are still not well understood. Fortunately, a few recent studies have devoted their efforts to investigating this subject. Rafael et al. [14] experimentally studied the impact of a single droplet on a moving liquid film and identified four regimes: tranquil coalescence, violent splashing, partial and complete bouncing, and surfing, which can be determined by the ratio of the droplet and surface velocities, and the liquid properties. Mitchell et al. [15] also conducted experiments to investigate a single droplet impact on a moving liquid film. They observed the crater formation and propagation, and presented that the crater area for the moving film increases similarly to that for the static film during the initial impact stage, while the crater shape itself is less similar and is asymmetrical due to the film motion. Cheng et al. [16] used a lattice Boltzmann method (LBM) to investigate the crown evolution during a single droplet impact on a moving liquid film. The film velocity was found to be an important factor affecting the splash, and the critical threshold of splash depends on Reynolds number and Weber number. Raman et al. [17] conducted LBM simulations to study the effects of the separation gap between the two droplets, film thickness, film velocity, liquid viscosity, and gas density on the crown evolution during two droplets impact on a moving liquid film. The simulations demonstrated that the crown structure and the central uprising jet are influenced by all these factors.

It should be noted that only impact dynamics were focused on in Refs. [14–17]; however, heat transfer characteristics were not taken into account on basis of the assumption that the droplet, liquid film, and wall have the same temperature. A very recent experimental study [18] indicated that when a water droplet impacts a thin heated wafer on which a moving cold water film is pregenerated, transient responses of the temperature and heat transfer coefficient show clear asymmetry, which differs from those observed on the stationary liquid film significantly. Thus, it is necessary to implement more studies to understand the variation of heat transfer characteristics when droplets impact a moving liquid film. In this work, a CLSVOF method is employed to investigate the impact of a double droplet on a moving wall with a preformed water film. The impact hydrodynamics and local heat transfer characteristics are examined for various film moving velocities, horizontal spacings between the two droplets, impact velocities, and film thicknesses.

2. Numerical method

2.1. CLSVOF method

The VOF method proposed by Hirt and Nichols [19] is employed to simulate the transient process of a double droplet impacting a liquid film. Its main idea is to capture a liquid-gas interface by defining a fluid volume function F, as follows,

$$F(x,y,t)=\begin{cases}0 & \text{gas}\\ 0<\alpha<1 & \text{two phase fluid}\\ 1 & \text{liquid}\end{cases} \tag{1}$$

where α denotes the volume fraction of fluid in a calculation unit with $\alpha = 0$ for the gas phase, $\alpha = 1$ for the liquid phase, and $0 < \alpha < 1$ for the interface phase.

The governing equation of F can be expressed as,

$$\frac{\partial F}{\partial t}+\vec{V}\cdot\nabla F=0 \tag{2}$$

where $\vec{V}$ is the velocity vector and t is the time.

Once the distribution of F function is obtained, the location and shape of the interface can be determined. However, because of its discontinuity at the interface, it is difficult to accurately calculate the physical quantity related to the curvature.

Level-set method captures an interface by defining a distance function ϕ, as follows [20],

$$\phi(x,y,t)=\begin{cases}d(x,y,\Gamma(t)) & (x,y)\in\Omega_1\\ 0 & (x,y)\in\Gamma(t)\\ -d(x,y,\Gamma(t)) & (x,y)\in\Omega_2\end{cases} \tag{3}$$

where $\Gamma(t)$ is the isosurface of $\phi = 0$ which represents the interface, d is the distance to the interface, and Ω_1 and Ω_2 are the regions of gas and liquid phases.

The governing equation of ϕ is as follow,

$$\frac{\partial \phi}{\partial t}+\vec{V}\cdot\nabla \phi=0 \tag{4}$$

The normal vector $\vec{n}$ and the curvature κ for the interface can be calculated by,

$$\vec{n}=\frac{\nabla\phi}{|\nabla\phi|} \tag{5}$$

$$\kappa=-\nabla\cdot\frac{\nabla\phi}{|\nabla\phi|} \tag{6}$$

By introducing the smooth Heaviside function, the interface is assigned a fixed finite thickness with the order of ε, or

$$H(\phi)=\begin{cases}0 & \phi<-\varepsilon\\ \frac{1}{2}\left[1+\frac{\phi}{\varepsilon}-\frac{1}{\pi}\sin\left(\frac{\pi\phi}{\varepsilon}\right)\right] & |\phi|\leqslant\varepsilon\\ 1 & \phi>\varepsilon\end{cases} \tag{7}$$

where ε presents the interface numerical thickness, which is taken as 1.5 times grid spacing here. Thus, the density and viscosity can be calculated by,

$$\rho(\phi)=\rho_g+(\rho_l-\rho_g)H(\phi) \tag{8}$$

$$\mu(\phi)=\mu_g+(\mu_l-\mu_g)H(\phi) \tag{9}$$

where ρ is the density, μ is the viscosity, the subscripts g and l denote the gas and the liquid phase, respectively.