Applied Mathematical Modelling 62 (2018) 510–525

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

Numerical and experimental study of oscillatory behavior of liquid surface agitated by high-speed gas jet

Mohammad Adib[a], Mohammad Ali Ehteram[b], Hassan Basirat Tabrizi[a,*]

[a] Department of Mechanical Engineering, Amirkabir University of Technology, 424 Hafez Avenue, Tehran 15875-4413, Iran
[b] Mechanical and Energy Engineering Department, Shahid Beheshti University A.C., Tehran, Iran

ARTICLE INFO

Article history:
Received 25 November 2017
Revised 9 May 2018
Accepted 22 May 2018

Keywords:
Jet impingement
CFD simulation
Instability
Cavity oscillation
Peak frequency
Rising ligaments

ABSTRACT

This study examines high-speed air jet impingement on water surface in the penetration mode using CFD simulation and experimental observation. An axisymmetric approach was taken along with the volume of fluid (VOF) method and the $k-\omega$ SST turbulence model for simulations. Moreover, a test rig was designed and fabricated to diagnose the phenomenon by tracking the interface motion using a high-speed camera in conjunction with an image processing procedure, and to evaluate the validity of the numerical simulations. Important aspects of the oscillatory behavior of the interface, cavity depth oscillation and rising ligaments and sheets, are addressed in detail. The simulation results offer a very close prediction of the peak frequencies of the cavity oscillation, and the number of ejected liquid sheets and ligaments despite some discrepancies in the range of the cavity motion and the height of ligament and sheets. Induced waves on the interface, and bubbles escaping from the cavity, which are the main source of instability of this two-phase system, are reproduced successfully.

1. Introduction

Impingement of high-speed gas flow on an initially quiescent liquid pool deforms the interface, forms a depression, and creates waves that propagate along the interface. This mechanism is encountered in various industrial applications such as controlling the liquid film thickness in coating processes [1,2], reducing the carbon content in the metal bath [3], and immersion lithography [4]. The interface motion regime has been found to be largely determined by the configuration, flow, and physical parameters. Depending on these parameters three distinct regimes have been identified, namely dimpling, splashing, and penetrating. The dimpling regime is marked by only a small indentation on the liquid surface. Increase in the jet momentum or closer injection distances makes the indentation deeper and disturbs the interface to the extent that the liquid begins splashing, and ripples are thrown outwards. If the intensifying trend continues, the two-phase system reaches another critical state at which further increment in momentum or reducing injection distance creates a deeper cavity with strong oscillations. In addition, the upwards and radially inwards motion of splashes prevails over the previously predominant outward motion [5]. Determining the cavity's static (or average) features such as depth, width, and shape has been the main goal of the early researchers that inspected this phenomenon. These studies tended to establish an analytical model, which relates the features of cavity to momentum of jet via force balance at the stagnation point combined with a

* Corresponding author.
E-mail address: hbasirat@aut.ac.ir (H. Basirat Tabrizi).

https://doi.org/10.1016/j.apm.2018.05.031

dimensional analysis, while not taking into account the effects of surface tension and shear, and assigning a proper value to the proportionality factor(s) appeared in correlations based on the experiments [6–10]. Furthermore, energy balance was employed in some studies [11]. The accuracy of the proposed correlations was evaluated by Nordquist et al. [12]. Gaussian or parabolic curvature was often assumed for the cavity shape [13]; conformal mapping methods were also employed to estimate the profile of cavity [14,15] The onset of instability and ascertaining the critical conditions have also been a concern. Viscous effects, despite their insignificant role in cavity features, were identified as a key factor in the sputtering effect in the liquid [6]. Density difference was also shown to play a major role, while surface tension works as a stabilizing agent [16]. Shear effects due to the Kelvin–Helmholtz instability are responsible for the inception of waves at the impact point and spreading thereof to the entire interface [17]. As the cavity becomes unstable, bubbles escaping from the cavity are known to be more effective than the Kelvin–Helmholtz instability in the forming of these ligaments and sheets [18]. Stretched ligaments and sheets exhibit a similar behavior to the isolated filaments [19], which break up into droplets depending on the aspect ratio and liquid properties [20]. This droplet generation can be used as an atomization mechanism [21] and to enhance the efficiency of cyclone separators [22]. Separated drops that bounce back to the oscillating liquid surface have the potential to create further instabilities [23]. Moreover, the cavity begins oscillating in size and depth. The frequency of oscillation increases as the mode of surface deformation transitions from dimpling to splashing mode [24] , and reaches the highest value in the penetration mode [25].

Numerical simulation of the phenomenon has faced problems due to the complexities of interface tracking and including surface tension effects. The volume of fluid (VOF) method [26] has proved effective in this matter by virtue of simplicity of implementation and good mass conservation compared to other fixed grid methods [27]. Most of the recent literature studies have used this method except for some that solved the problem by using simplifying assumptions including static cavity profile [28], negligible surface tension and shear stress [29] or alternative methods such as level-set [30,31]. There are also precedents for using more complex two-fluid formulation (instead of single-fluid variable density approach) among earlier research [32,33].

Nguyen and Evans [34] performed an axisymmetric simulation using the standard $k-\epsilon$ turbulence model. Penetration of a supersonic air jet through water was investigated by Morshed et al. [35]. Solórzano-López et al. [36] attempted to observe and simulate the circulation mechanism induced in liquid by slanted jets. Muñoz Esparza et al. [37] focused on dimpling and splashing modes of a planer jet impingement on water, and compared the numerical results against Particle Image Velocimetry (PIV) data. Li et al. [38] examined a combined blower with different number of nozzles. Yannick et al. [39] suggested a zonal approach for description of a compressible gas jet. Reynolds [40] investigated the interaction between a plasma arc and molten material.

Although previous studies covered some features of this phenomenon, important aspects of the oscillatory behavior of liquid, specifically cavity oscillation, and ligament ejection, have not been subjected to a comparative numerical and experimental investigation. Moreover, the validity of computational models for prediction of these aspects has never been evaluated. This study presents a more in-depth examination of oscillatory behavior of the interface in the penetration mode of surface deformation numerically and experimentally with a specific emphasis on the frequency of cavity depth oscillation and characteristics of the ejected liquid ligaments and sheets.

2. Governing equations

The governing equations follow the assumptions and principles on which the VOF method is established, that is, a single set of equations for the continuity, momentum, and turbulence quantities is solved. In addition, the volume fraction of fluids is determined by solving a conservation equation. All equations are expressed in Cartesian coordinate system.

2.1. Continuity and momentum

The shared continuity and momentum equations for the multiphase system can be written as

$$\frac{\partial \rho u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i \tag{2}$$

where ρ is the density, p the pressure, u_i the ith component of mean velocity vector, g_i the ith component of the gravity force. τ_{ij} is the effective stress tensor which is defined as:

$$\tau_{ij} = (\mu + \mu_t)\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right] \tag{3}$$

where μ, δ the molecular viscosity, and Kronecker delta, respectively. In Eq. (3), μ_t is turbulent viscosity, which is determined by turbulence model. Here $k-\omega$ SST [41] is employed for turbulence closure.