International Communications in Heat and Mass Transfer 75 (2016) 282–294

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

# Application of a two phase lattice Boltzmann model in simulation of free surface jet impingement heat transfer☆

H. Amirshaghaghi [a], M.H. Rahimian [a,*], H. Safari [b]

[a] School of Mechanical Engineering, College of Engineering, University of Tehran, Iran
[b] Institute for Computational Modeling in Civil Engineering, Technische Universität Braunschweig, Braunschweig, Germany

ARTICLE INFO

Available online 29 April 2016

*Keywords:*
Liquid jet impingement
Free surface
Two phase lattice Boltzmann model
Heat transfer

ABSTRACT

In this paper a two-phase lattice Boltzmann model, capable of handling large density jumps, is used to study the free surface jet impingement cooling in the non-boiling regime. The multiple-relaxation-time (MRT) collision operator is employed to enhance the numerical stability of the model at high Reynolds numbers. The capability of the outlined two-phase LB scheme in accurate capturing of the interface shape is assessed through relevant test cases. These include the oscillating drop, Rayleigh–Taylor instability, and Kelvin–Helmholtz instability. In addition to demonstrate the validity of the model in simulation of heat transfer the predicted distribution of the Nu number on the impingement plate are compared with those of analytical results. The validated numerical solver then is employed to study a planar liquid jet emanating from a nozzle into a quiescent gas and impinging the opposite plate. The influence of the jet Re number and liquid-to-gas density ratio on the instantaneous flow field, interface shape, and heat transfer rate is also examined. This work reports for the first time the simulation of an impinging free surface liquid jet using a two phase lattice Boltzmann model.



## 1. Introduction

Liquid impinging jets are used in numerous industrial applications such as cooling of electronic devices (power electronics) and annealing of hot metal sheets. This extensive use is because of their highly favorable heat transfer characteristics and relatively simple implementation. Compared to other heat transfer arrangements liquid impinging jets provide much more efficient use of flow and higher convective heat transfer rates. For example, due to very thin boundary layer, liquid impingement flow offers very low thermal resistance and can remove heat loads as high as 400 MW/m2 [1].

Based on the medium that the jet travels through, there are two common liquid jet configurations: the free-surface jet (unsubmerged jets), in which the liquid is exhausted into a less dense gaseous medium; and the submerged jet, in which the fluid is injected to the same medium. Further, generally two types of jet configurations are employed, that is circular and slot (planar) jets. The jets of interest in the present paper are the planar free surface jets.

Free surface impinging liquid jets are widely studied experimentally [2–8]. However, due to multitude of affecting parameters such as nozzle shape and size, location of exhaust ports, nozzle-to-target spacing, surface motion, jet velocity and temperature, liquid properties, and critical heat flux, one frequently encounters technical barriers to obtaining accurate experimental measurements. Therefore, developing validated theoretical models is a powerful alternative to experimental investigations.

A variety of analytical solutions and numerical simulations of heat transfer in free surface planar jet impingement can be found in literature. Levy [9] numerically solved hydraulic and thermal boundary layer equations for the Falkner–Skan flow over isothermal and isoflux surfaces. It was indicated that when the interior wedge angle is π the heat transfer coefficients are identical over constant temperature and constant heat flux surfaces. Inada and Miyasaka [10], proposed a semi-analytical solution for the flow and heat transfer in a two-dimensional impinging jet of water. They compared the obtained results for local heat transfer distribution on the isoflux flat plate with experimental ones and examined the effect of changing the nozzle distance to nozzle diameter ratio on heat transfer coefficients. Chen et al. [11] conducted a theoretical analysis to study the heat transfer characteristics under normally impinging slot jets. They divided the heat transfer surface into four regions and provided a general description to heat transfer rate in each zone. Although their model was originally developed for laminar flows, it was shown that even at high Reynolds numbers; the model predicts the thermal characteristics with a reasonable accuracy.

It should be noted that despite favorable features of analytical solutions, there are several major shortcomings related to use of this class of solutions. The major problems can be summarized as limited range of validity, inaccuracy due to oversimplification of geometry and flow field, inability in capturing flow unsteadiness, and ignoring the presence of gas phase.

☆ Communicated by Dr. W.J. Minkowycz
* Corresponding author.

http://dx.doi.org/10.1016/j.icheatmasstransfer.2016.04.020

To alleviate these issues several studies are devoted to efficient numerical simulations of the impinging free surface liquid jets. Note that in order to obtain accurate predictions of liquid jet behavior it is crucial to use proper schemes to compute the interface dynamics. There are several interface localization methods such as continuum approaches with front tracking [12] or front capturing methods including Volume of fluid (VOF) and Level-set (LS) [13,14]. Tong [15] numerically studied the impingement of a free surface oblique water jet on heated plate. The Navier–Stokes equations were solved using a finite-volume formulation and the interface was tracked by VOF method. Results quantitatively verified with available experimental and analytical data. Rohlfs et al. [16] investigated the influence of Reynolds number, Prandtl number, and the geometrical parameters on the heat transfer from a flat plate during the impact of a laminar free surface jet. In their study the VOF formulation was implemented to approximate the free surface. Based on the simulation results, the authors proposed a new dimensionless time scale. This non-dimensional quantity was then employed to establish a new correlation for stagnation zone heat transfer that covers a wide range of operating conditions. Lewis et al. [17] also used the VOF method to study the cooling behavior of the impingement of circular liquid jets as well as a train of droplets. The obtained results for the liquid jet verified with Falkner–Skan wedge flow predictions. It was shown that the velocity profile at nozzle exhaust has a significant impact on the heat transfer coefficients. Further, it was pointed out that both jet configurations exhibit superior cooling performance compared to the train of droplets.

It should be noted that the accuracy of the aforementioned studies is limited to the second order accuracy of implemented VOF model. A consequence can be some uncertainties in the shape of interface and thus on calculation of interface curvature and surface tension forces. Besides, the VOF computations are demanding in terms of computer time and memory, which put some constraints on use of VOF method. The LS method reduces the problem found with VOF method since the interface location and curvature are readily accessible from the iso-level smooth function. Son et al. [18] employed a level set approach to localize the liquid/gas interface. They examined the turbulent liquid impingement on a moving plate and conducted a parametric study to evaluate the various fluid flow design scenarios. Needless to say, the level set approach has its own drawbacks. Most importantly, the level-set function inherently lacks conservation properties. Various approaches have been proposed to reduce this issue such as the coupled VOF-LS model of Sussman and Pucket [19]. While this method has been quite successful, it suffers from problems such as implementation complexity, higher computational cost than simple LS method, and time step size restrictions [20].

As a summary one may conclude that despite the growing body of research, the accurate simulation of free surface liquid jet impingement using the classical approaches is still a very challenging task. The major problems consist of tedious numerical algorithms, high computational costs, uncertainties in the interface shape, and intricate handling of sharp discontinuities at the interface as well as the lack of mass conservation properties.

In recent decades, the lattice Boltzmann method (LBM) has emerged as an alternative to Navier–Stokes based solvers. Due to its mesoscopic nature, LBM is a promising tool to cope with interface of non-ideal gases and binary fluids [21]. Multiple approaches have been discussed in literature to develop appropriate models for multi-phase flows in lattice Boltzmann framework. Examples include the early work of Gunstensen et al. [22], Shan and Chen method [23] based on microscopic interaction of particles [23], free energy based model of Swift et al. [24], and He et al. [25] approach.

The main drawback of these approaches and similar studies [26–28] is mostly regarding the numerical instability at moderate and high density ratios as well as the high Reynolds numbers. Moreover, the majority of these models suffer from parasitic currents that plague the flow field. These issues restricted the applicability of LB method in simulation of realistic systems such as free surface liquid jets, which normally occurs at moderate and high density ratios. In order to alleviate these problems, Lee and Lin [29] proposed a new formulation, using a two-distribution discrete Boltzmann equation (DBE). Resorting to a collection of consistent discretization strategies and proper treatment of surface tension force, they stabilized otherwise unstable LBE formulation at high density ratios. Further, to enhance the numerical stability of LBM at high Reynolds numbers D'Humieres [30] proposed the idea of using multiple-relaxation-time (MRT) instead of a single time scale in calculation of collision term. It was shown that comparing to single-relaxation-time (SRT) the MRT technique is more successful in damping the high frequency oscillations of traveling waves generated by dynamic pressure. Fakhari and Lee [31] extended this approach to two-phase LBM and studied the evolution of shear layer for two immiscible fluids, also known as Kelvin–Helmholtz instability, with density contrast at Reynolds numbers up to 10,000. The Kelvin–Helmholtz instability (KHI) is an important shear instability mechanism or two phase flows. In particular, except the Rayleigh mode where the capillary instability is predominant, the KHI along with Rayleigh–Taylor instability is thought to play the leading role in the shape of liquid/gas interface [32]. Hence, proper simulation of the KH instability is crucial to accurate predictions of liquid jet breakup. The significance of Fakhari and Lee's study lies here. Since their two-phase MRT-LB model is capable of capturing the shear instabilities and accurate prediction of interface shape at high Reynolds numbers, it is speculated to be a reliable framework for simulation of liquid jets exhausted into a gaseous atmosphere. However, this conjecture is still to be verified, using the simulations of the current study.

It is interesting to mention that, to the knowledge of authors, no study has specifically addressed the simulation of free surface liquid jet impingement using LB method. The relevant publications deal with theoretical aspects of applying the LBM to free surface flows [33–35]. In this class of solutions; in contrast to multiphase models; the capillary forces are usually neglected and the lattice Boltzmann equations are only solved in the nodes of the dense phase. Note that such a formulation reduces the computational costs and leads to reasonable accuracy at high density and viscosity ratios. But, in the case that either the liquid-to-gas density ratio is below 500 (and thus aerodynamic forces are important) [36] or when one phase is dispersed into the other (e.g. gas bubble entrapment) the approach will not be reliable.

In the present study, considering the problems and drawbacks associated with implementation of classical approaches as well as the LB/free surface models, we focus on validation and application of binary (two-phase) LBM flow solver to simulation of free surface liquid jet impingement. Using the validated code the hydrodynamic and thermal characteristics of impinging slot liquid jet introduced into a lighter phase is examined. In addition, the effect of jet inlet velocity and density ratio on the flow field and heat transfer rate is investigated.

In the sections that follow, first the theoretical framework is described. In Section 3, several relevant benchmarks are tested to validate the theoretical framework. In Section 4 the computational results for an impinging liquid jet exhausted into a gaseous atmosphere is presented and effect of several parameters on the flow field and heat transfer is examined. Concluding remarks are given in Section 5.

## 2. Mathematical modeling and governing equations

### 2.1. Computation of the flow field using LBM

Discrete Boltzmann equation (DBE) of a non-ideal fluid with a generalized collision operator $\boldsymbol{\Lambda}$and intermolecular forcing term is written as

$$\frac{Df_\alpha}{\mathrm{Dt}} = \frac{\partial f_\alpha}{\partial t} + \mathbf{e}_\alpha \cdot \nabla f_\alpha = -\boldsymbol{\Lambda}\left(f_\alpha - f_\alpha^{\mathrm{eq}}\right) + \frac{1}{\rho c_s^2}\left(\mathbf{e}_\alpha - \mathbf{u}\right) \cdot \mathbf{F} f_\alpha^{\mathrm{eq}} \quad (1)$$