International Journal of Heat and Mass Transfer 125 (2018) 876–890

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Pool boiling simulation using an interface tracking method: From nucleate boiling to film boiling regime through critical heat flux

Yohei Sato *, Bojan Niceno

Paul Scherrer Institute, 5232 Villigen PSI, Switzerland

ARTICLE INFO

Article history:
Received 16 February 2018
Received in revised form 23 April 2018
Accepted 25 April 2018

Keywords:
Computational fluid dynamics
Interface tracking method
Pool boiling
Nucleate boiling
Film boiling
Critical heat flux

ABSTRACT

A series of pool boiling simulations has been performed using an interface tracking method in the framework of Computational Fluid Dynamics (CFD). The boiling regime simulated in this work covers from the nucleate boiling to the film boiling through Critical Heat Flux (CHF). The numerical method previously developed by the authors (Sato, Niceno, Int. J. Heat Mass Trans., 105, 505–524) is used; the liquid-vapor interface is resolved by a color function, and conjugate heat transfer between the wall and the fluid is included. The micro-layer, which is the thin liquid film existing beneath a growing bubble, is taken into account using a specialized model. The validation case is the experiment by Gaertner: the boiling of water from a heated, horizontal plate under atmospheric pressure. The applied heat flux ranges from 50 to 1500 kW/m^2. The computed heat transfer coefficient agrees well with the measured value, demonstrating the capability of the described simulation method to predict boiling heat transfer for the wide range of boiling regimes.

1. Introduction

The prediction of CHF is one of the most important research topics in the field of heat and mass transfer because higher heat flux than CHF results in burnout in heat-transfer machineries. However, the prediction of CHF still relies heavily on empirical data, since direct numerical analysis/simulation of the boiling process, even using advanced modeling techniques such as CFD, are not yet sufficiently mature to guarantee trustworthy results in all conditions. In this paper, we try to overcome this situation, i.e. to compute CHF by using one of the most advancing CFD simulation methods and a powerful supercomputer.

Since vaporization takes place at the liquid-gas interface, it is essential to take into account the interface shape and area in the boiling simulation. Thus, numerical approaches which can resolve the interface are considered to be appropriate for the simulation with phase change phenomena. Most of all the interface resolving approaches developed for two-phase flow have been applied to boiling simulations, e.g. arbitrary Lagrangian-Eulerian approach [1–3], level set method [4,5], volume of fluid method [6,7], color function method [8–10], front tracking method [11], phase field method [12–14] and lattice Boltzmann method [15–17]. Using these approaches, nucleate boiling or film boiling simulations have been successfully performed as reviewed in [18]. However, the boiling simulation at high heat flux, e.g. CHF, is rare because (i) the flow field becomes rough due to high mass-transfer rate which introduces numerical instability and (ii) appropriate nucleation site model is required for the heat-transfer surface applied with such a high heat flux.

A nucleate pool boiling simulation near CHF was performed by Son and Dhir [19], in which multiple nucleation sites were incorporated, and the level set method was used. Due to the restriction imposed by the constant-temperature assumption over the heat-transfer surface, the bubble waiting time could not be computed, and had to be prescribed *a priori*. Nonetheless, the authors were able to obtain acceptable results for nucleate pool boiling of water for applied heat fluxes up to 800 kW/m^2 at atmospheric pressure. The validation case is the measurement done by Gaertner [20] and the CHF measured in the experiment was around 1000 kW/m^2.

Li et al. [16], and Gong and Cheng [17,21] have simulated from the nucleate boiling to film boiling regimes through CHF using lattice Boltzmann method. The influence of the wettability of the heat-transfer surface on the heat transfer coefficient was evaluated. However, the computation is limited to two dimensions, and no validation was reported.

Using an sharp-interface phase-change model [8] and a depletable micro-layer model [22], present authors have simulated the pool boiling of Gaertner's experiment [20] from the discrete bubble regime (50 kW/m^2) to the vapor mushroom region (300 kW/m^2)

* Corresponding author.
E-mail addresses: yohei.sato@psi.ch (Y. Sato), bojan.niceno@psi.ch (B. Niceno).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.04.131
0017-9310/

which is published in [23]. Unlike the simulation done by Son and Dhir [19], the temperature of the heat-transfer surface is not prescribed in order to compute the bubble waiting time based on the calculated wall temperature at each nucleation site. The computed heat transfer coefficient was in good agreement with experimental data, though the applied heat flux was lower than CHF ($\approx$1000 kW/m^2).

The depletable micro-layer model [22] was developed by us in order to take into account the thin liquid film existing beneath a growing bubble in nucleate boiling, which can be measured in experiments of boiling water at atmospheric pressure by using optical techniques [36–38]. The micro-layer is considered to be formed because the growing vapor bubble cannot exclude the liquid on the wall while the bubble grows faster in the bulk due to less constraint. In order to understand the conditions required for forming micro-layer, Fischer et al. [39] performed an experiment using a refrigerant as the working fluid. They concluded that the velocity of the liquid-vapor interface, the wall superheat and the latent heat of evaporation have a major influence on the forming or non-forming of the micro-layer. Recently, Urbano et al. [40] developed Direct Numerical Simulation (DNS) method for nucleate boiling in the axisymmetric coordinate system. They derived an interesting result; micro-layer is formed if the growth of the bubble exceeds a limit velocity, otherwise it is not formed. The limit velocity found to depend on the thermos-physical properties of the fluid, and corresponds to a combination of the dewetting velocity and the velocity locally induced by the phase change at the contact line.

In this paper, we apply the simulation method [23] to a pool boiling simulation from the nucleate boiling to film boiling regime through CHF. Again, the saturated pool boiling experiment of Gaertner [20] has been selected as a suitable validation test. The working fluid is water at the atmospheric pressure, and a horizontal heat-transfer surface is used. In the simulation, the dimension of the heater block is the diameter of 20 mm with the thickness of 6 mm. A constant heat flux is imposed to the bottom of the heater block; the applied heat fluxes are 50, 100, 300, 600, 800, 1000, 1200 and 1500 kW/m^2, which cover the boiling regime from the nucleate boiling to the film boiling. Each simulation is continued until saturated conditions have been attained. Since the heater power controlled system is employed in both the experiment and the simulation, the transition boiling cannot be treated in this study. Note that the transition boiling can be observed only when the wall superheat increases accompanied by a reduction of the heat flux [24]. The supercomputer at the Swiss National Supercomputing Center CSCS (a maximum capability of 1.3 petaflops) was used to accelerate the simulations.

The structure of the paper is as follows: in Section 2, the numerical approach is briefly explained. The validation of the simulation method is described in Section 3, where simulation/measurement results are compared. Conclusions from the study are drawn in Section 4.

2. Numerical method

The numerical method used in this paper is identical to the one proposed by present authors in [23], and it is briefly explained here. Note that the main difference between the this paper and the previous one [23] is the boundary condition, e.g. the range of applied heat flux and the geometry of heater, and the computational domain size.

2.1. Navier-Stokes solver

The incompressible Navier-Stokes equations are solved using a finite-volume approach on a fixed, rectangular, Cartesian grid using the projection method [25]. The governing equations for mass and momentum conservation is written as follows:

$$\nabla \cdot \vec{u} = \left(\frac{1}{\rho_v} - \frac{1}{\rho_l}\right)\dot{m}, \tag{1}$$

$$\frac{\partial(\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u}\,\vec{u}) = -\nabla p + \nabla \cdot \left\{(\mu + \mu_t)(\nabla \vec{u} + (\nabla \vec{u})^T)\right\} + \vec{f}, \tag{2}$$

where $\vec{u}$ (m/s) is the velocity vector, ρ (kg/m^3) the density, the subscripts l and v denoting the liquid and vapor phases, respectively, $\dot{m}$ (kg/m^3 s) the phase change rate, t (s) the time, p (Pa) the pressure, μ and μ_t (Pa s) are the dynamic and turbulent eddy viscosity, respectively, and $\vec{f}$ N/m^3) the body-force vector. In the convention adopted here, the phase change rate $\dot{m}$ is positive for vaporization, and negative for condensation. Turbulence is modelled with Large Eddy Simulation (LES) using the Smagorinsky subgrid-scale model [26], the Smagorinsky constant being set to $C_s = 0.17$ in this work. The projection method [25] is used for the solution algorithm for pressure-velocity coupling.

Using the color function ϕ to signify the volume fraction of liquid inside any control volume, i.e. a grid cell of the underlying mesh, the average density and viscosity within the volume are respectively defined as follows (i.e. via linear interpolation):

$$\rho = \phi\rho_l + (1-\phi)\rho_v \text{ and } \mu = \phi\mu_l + (1-\phi)\mu_v. \tag{3}$$

A mass-conservative, interface-tracking method based on the color function approach [27] is employed to delineate the boundaries between the liquid and vapor phases. The transport equation for the color function can be written as:

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \vec{u}) = -\frac{1}{\rho_l}\dot{m}, \tag{4}$$

with the rational CIP-CSL2 scheme [27,28] being adopted for the computation of the advection term. To prevent smearing of the color function, an interface sharpening algorithm [29] is needed. The coupling between the sharpening scheme for the color function used here and the contact angle treatment of the CSF model is described in detail [30].

The thermal energy balance equation is solved under the assumption that the temperature at the liquid-vapor interface is constant, and equal to the saturation temperature, T_{sat}. The governing equation can then be written as follows:

$$C_p\left(\frac{\partial T}{\partial t} + \vec{u} \cdot \nabla T\right) = \nabla \cdot ((\lambda + \lambda_t)\nabla T) + Q, \tag{5}$$

where C_p (J/m^3 K) is the specific heat at constant pressure, T (K) is the temperature, λ (W/km) the thermal conductivity, λ_t (W/km) the turbulent thermal conductivity, and Q (W/m^3) the volumetric heat source. The turbulent thermal conductivity is calculated according to

$$Pr_t = \frac{\mu_t/\rho}{\lambda_t/C_p}, \tag{6}$$

where $Pr_t (= 0.9)$ is the turbulent Prandtl number, assumed to be constant in this study.

The computational domain consists of both fluid (liquid and vapor) and solid sub-domains, the solid sub-domain being identified using the immersed boundary method [31,32]; conjugate heat transfer between the solid and the fluid is fully taken into account.