Journal of Computational Physics 300 (2015) 20–52

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A depletable micro-layer model for nucleate pool boiling

Yohei Sato *, Bojan Niceno

Paul Scherrer Institute, Villigen PSI, 5232 Switzerland

ARTICLE INFO

Article history:
Received 21 December 2014
Received in revised form 4 July 2015
Accepted 24 July 2015
Available online 30 July 2015

Keywords:
Phase change
Interface tracking
Micro-layer model
Nucleate boiling
Conjugate heat transfer

ABSTRACT

A *depletable micro-layer* model has been developed for the simulation of nucleate pool boiling within the framework of Computational Fluid Dynamics (CFD) modeling using an interface-tracking method. A micro-layer model is required for the CFD simulation to take into account vaporization from the thin liquid film – called the micro-layer – existing beneath a growing vapor bubble on a hot surface. In our model, the thickness of the micro-layer is a variable defined at each discretized fluid cell adjacent to the heat-transfer surface; the layer decreases due to vaporization, and can finally disappear. Compared to existing micro-region models, most of them based on the concept of contact-line evaporation, as originally proposed by Stephan and Busse, and by Lay and Dhir, our model incorporates simplified modeling ideas, but can nonetheless predict the temperature field beneath the growing bubble accurately. The model proposed in this paper has been validated against measurements of pool boiling in water at atmospheric pressure. Specifically, the bubble principal dimensions and the temperature distribution over the heat-transfer surface are in good agreement with experimental data.

1. Introduction

Nucleate boiling heat transfer is one of the most efficient modes of heat transfer under high heat-flux conditions, and is used in many engineering applications, such as refrigeration, cooling systems, boilers, heat exchangers, etc. Although there is a multitude of applications, accurate prediction of boiling heat transfer still relies heavily on empirical methods, because direct numerical simulation, including Computational Fluid Dynamics (CFD), is not yet sufficiently mature to produce reliable results. The first work of CFD simulation dealing with bubble growth in nucleate boiling was reported by Lee and Nydahl [1]. Here, the bubble motion and growth were captured using a moving mesh system (i.e. with the bubble surface coinciding with a grid line) in cylindrical coordinates. Although a relatively coarse mesh was adopted around the bubble surface, the computed bubble growth showed good agreement with experimental data. Following this work, a variety of phase-change models for CFD have been developed. These often involve the concept of "direct interface tracking". Examples are the Level Set (LS) method [2–6], the front-tracking method [7–10], the Volume Of Fluid (VOF) method [11–21], the coupled LS and VOF method [22], and phase-field [23–25] approaches. The current state-of-the-art on this subject is that phase-change phenomena, especially those pertaining to unbounded media, can be simulated accurately using CFD [26]. However, in the case of nucleate boiling, a satisfactory numerical treatment of the thin liquid film existing beneath the growing bubbles on the heat-transfer surface still appears to be lacking.

* Corresponding author. Tel.: +41 (0) 56 310 26 66; fax: +41 (0) 56 310 44 81.
E-mail address: yohei.sato@psi.ch (Y. Sato).

http://dx.doi.org/10.1016/j.jcp.2015.07.046
0021-9991/

The existence of this thin liquid film, called the *micro-layer*, was confirmed experimentally using optical interferometry by Sharp [27] and Jawurek [28] in the 1960s. These pioneering works were followed by several others [29–31], all with similar conclusions. These works revealed that the thickness of the micro-layer is of the order of few micrometers. Massive phase-change processes take place at the upper surface of the micro-layer, driven by the high heat flux through the layer. The high temperature gradient associated with this heat flux is the result of: (i) the thinness of the layer, and (ii) the large temperature difference between the top and the bottom of the layer, the latter in touch with the heat-transfer surface. Recent measurements for boiling water at atmospheric pressure [32] indicate that the heat flux through the micro-layer can exceed 1 MW/m^2. Consequently, a heat flux of this magnitude needs to be taken into consideration in numerical simulations if accurate prediction of the overall heat transfer process is to be achieved. In the case of nucleate pool boiling of water at atmospheric pressure, a growing vapor bubble typically occupies a volume just a few millimeters in diameter, while the computational domain required for CFD would need to be of the order of a few centimeters to capture the flow around a growing/rising bubble. In short, a computational cell used in a CFD simulation would be typically much larger than 50 μm, whereas the micro-layer thickness is of the order of few micrometers. On this scale, the flow inside the micro-layer cannot be computed directly due to lack of spatial resolution, and some modeling approach for the micro-layer, a so-called micro-layer or micro-region model, is required.

The modeling of the formation of this micro-layer was first attempted by Cooper and Lloyd [33]. Based on their careful experiment, the authors were able to derive an equation for the initial thickness of the micro-layer, δ_0, which turned out to be proportional to the square root of time from initiation. The micro-layer model of Cooper and Lloyd was introduced into a CFD simulation by Lee and Nydahl [1], and they estimated that 87 percent of the wall heat transfer is caused from the evaporation of micro-layer. An alternative model was proposed by van Stralen et al. [34], in which δ_0 depends on the interface velocity, and distance from the nucleation site. Note that these empirical models are based on the value of δ_0, which was not measured directly, but inferred from the measured temperature field. More recently, Utaka et al. [31] succeeded in measuring the micro-layer thickness directly using an He–Ne laser, and proposed a simple expression for δ_0 proportional to the distance from the nucleation site. Very recently, based on measurements from their own experiment [31], Chen and Utaka [35] proposed a mechanistic nucleate boiling simulation model, coupled with VOF, in which the initial thickness of the micro-layer δ_0 is given, but the transient thickness $\delta(t)$ is calculated explicitly. In this model, empirical data is required for the contact angle (i.e. the time history of the apparent contact angle must be prescribed), under the assumption of a spherical bubble shape. This prescription of the contact angle limits the applicability of the simulation model to those strictly pertaining to the measured conditions. Validation of the model is carried out under the assumption of cylindrical symmetry; good agreement is obtained for the bubble growth rate, but details of the temperature field are missing.

CFD simulation of nucleate boiling has been studied extensively by Dhir's group [36]. Their model features a micro-region model [37], in which disjoining pressure, vapor recoil pressure, and interfacial heat transfer resistance, are all taken into account. On the basis of a steady-state assumption, it is shown that the micro-region model may be simplified to a set of fourth-order Ordinary Differential Equations (ODEs). A similar micro-region model, but one neglecting the recoil pressure, was proposed by Busse and Stephan [38], which they coupled to a CFD code for their nucleate boiling flow simulations [39–41]. Using such micro-region models, it was demonstrated that nucleate boiling flows, including both pool boiling [21,39,40,42–45] and convective boiling [46–49], could be simulated successfully. Overall, the computed bubble growth rates are in good agreement with experimental measurement. However, there are several issues requiring closer examination: (i) the models themselves, (ii) the underlying assumptions used, and (iii) the parameters adopted in the models.

With regard to the modeling, the most contentious issue is that the micro-layer is modeled as a *line* and not as a *layer*; that is, the micro-region models [37,38] are categorized in terms of a contact-line evaporation model. Referring to Fig. 1(a), these micro-region models are effective only within the cell that incorporates the *triple line*; i.e. the line in which the liquid, vapor and solid phases jointly come into contact, and where the computed temperature profile over the heat-transfer surface always displays a sharp drop [39]. Although the computed temperature profile was not directly compared with measured data here, a typical data measurement, Fig. 1(b), displays an obviously different feature in the temperature profile on the wall [50]. The lower temperature region is that beneath the bubble (Fig. 1(b)), and not at the contact line, as predicted by the contact-line evaporation model (Fig. 1(a)). In order to reproduce such a temperature profile numerically, it appears to us that the micro-layer should be modeled explicitly as a layer, and not just as a line at the contact point. Another issue pertinent to the modeling approach is that the derivation of the ODEs remains fundamentally two-dimensional, or axisymmetric, and a three-dimensional formulation has yet to be established. In addition, at present, such models cannot handle contact angles larger than 90°, due to the limitation imposed by the lubrication theory embedded in them [37,38].

Another issue, relating to the assumption that the steady-state condition may be applied to the formulation, needs also to be examined. Note that steady-state in this context relates to the fact that the profile of the micro-layer thickness remains constant, even as the location of the triple line moves. In addition, the wall temperature over the heat-transfer surface, T_w, is assumed constant. Note that T_w can only be calculated if a quasi-steady assumption is employed [39,40].

Another important issue concerns the parameters used in the models: the value of the dispersion constant (Hamaker constant) is either unknown, or not well-established. For example, a negative value, which may be unphysical, is employed in [47]. Moreover, uncertain initial/boundary conditions have been employed for the integration of the ODEs: for example, the adsorbed film thickness, and the gradient of the film thickness at the extremities of the micro-region, have not been measured.