Journal of Computational Physics 375 (2018) 717–746

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Direct numerical simulations of incompressible multiphase magnetohydrodynamics with phase change

Jie Zhang [a], Ming-Jiu Ni [a,b,*]

[a] State Key Laboratory for Strength and Vibration of Mechanical Structures, School of Aerospace, Xi'an Jiaotong University, Xi'an, Shaanxi 710049, China
[b] School of Engineering Science, University of Chinese Academy of Sciences, Beijing 101408, China

ARTICLE INFO

Article history:
Received 29 October 2017
Received in revised form 3 August 2018
Accepted 1 September 2018
Available online 6 September 2018

Keywords:
Magnetohydrodynamics
Multiphase flows
Cut-cell method
Phase change

ABSTRACT

A new phase change model has been developed for the simulation of incompressible multiphase magnetohydrodynamics based on the Volume-of-Fluid method. To decrease the pressure oscillations when large density contrasts are present between the liquid phase and the vapor phase, a smooth distribution of sharp mass transfer rate within a narrow region surrounding the interface is adopted, and a ghost-cell approach is used to impose the saturating temperature at the liquid–vapor interface when solving the energy equation. After that, the method has been implemented in an incompressible multiphase magnetohydrodynamics solver developed in our previous work (Zhang and Ni (2014) [3]). Moreover, when computing the electromagnetic fields, a cut-cell approach is implemented to keep the sharpness of the interface, which is treated as an electrically insulating boundary as it translates and deforms with the fluid. The phase change model has been verified for a series of one-dimensional, two-dimensional and three-dimensional problems, while the numerical results agree well with either the theoretical solutions or the experimental data. In particular, by simulating the vapor bubble rising in superheated liquid under nonzero gravity in presence of external magnetic field, the magnetohydrodynamics effect on the vaporization of the rising bubble is investigated and we observe the magnetic fields to suppress the vapor bubble growth during the phase change. At last, both two-dimensional and three-dimensional film boiling simulations are conducted, which show good qualitative agreement with heat transfer correlations, and the vapor bubble is found to elongate along the direction of the magnetic field during its growth, moreover, the time instant for the vapor bubble to detach from the film is also delayed.

1. Introduction

As a contactless method, the magnetic fields are always employed to control the liquid metal flow in the metallurgy processes, for stirring or homogenizing purpose such as injecting the bubbles to refine the melt [1]. Besides, in fusion nuclear reactors, where liquid lithium or lithium–lead are widely used as a breeder and heat remover, the multiphase flows of the liquid metal are always subject to the strong magnetic fields [2], and thus their flow characteristics are greatly

* Corresponding author at: School of Engineering Science, University of Chinese Academy of Sciences, Beijing 101408, China.
E-mail address: mjni@ucas.ac.cn (M.-J. Ni).

https://doi.org/10.1016/j.jcp.2018.09.001
0021-9991/

affected by the magnetohydrodynamics (MHD, hereafter) effect. For this purpose, a numerical methodology to simulate the incompressible multiphase MHD flows has been proposed in our previous study [3], and the approach is developed as an extension of the open-source software Gerris flow solver [4], which combines the Volume-of-Fluid (VOF, hereafter) method and the adaptive mesh refinement (AMR, hereafter) technique for interface tracking and advancing. In that work, a consistent and conservative scheme [5,6] is introduced to solve the electromagnetic field, including the calculation of the electric potential and the Lorentz force. After that, several direct numerical simulations have been conducted to study the multiphase MHD flows, particularly of the single bubble motion [7–9] and the droplet impacting [10,11] problems under the influence of external magnetic fields.

However, another type of multiphase MHD flows involving the vaporization of the liquid metal still requires further investigation, as to achieve high thermal performance at high power density in the thermonuclear fusion device, in which the MHD effect always suppresses the heat transfer efficiency. As a consequence, the Evaporation of Lithium and Vapor Extraction (EVOLVE) blanket concept uses the vaporization of lithium to remove the blanket thermal power [12,13]. In addition, for the boiling lithium blanket, the evaluation of the impacts of magnetic field on various stable-boiling regimes is also in demand [13]. However, difficult as the experiments are [14–16], comprehensive information of the flow field is hardly obtained due to the opacity of the liquid metal, and moreover, the available experimental results are always inconsistent that no universal conclusion with respect to the influence of the magnetic field is obtained. Alternatively, direct numerical simulations would be an effective method to predict the flow behaviors of the phase change problems under the influence of magnetic fields. However, as far as the author know, none of such numerical methods has been proposed in the available publications.

To develop such a robust and accurate phase change model, the difficulties lie in two aspects. First, the mass transfer only happens in the mixed cells which contain the liquid–vapor interface, leading a discontinuity of the velocity field across the interface, and such singularity of the mass source probably produces numerical instability during solving the velocity–pressure coupling equations in vicinity of the interface, particularly with large density contrasts between the two phases. By using a VOF method, Hardt and Wondra [17] employed a smoothing function to smear the mass flux over 4–5 computational cells surrounding the interface, and then the distribution of the mass transfer rate was again artificially shifted to the liquid or vapor side, by this a very stable performance was obtained although the physical significance was declined. Furthermore, in merit of its ease of implementation, Kunkelmann and Stephane [18] further implemented this numerical scheme into the open source code OpenFOAM and then it was widely used. Magnini et al. [19] used a similar smoothing function to smear the mass source into the neighbor cells and the bubble motions during flow boiling in a microchannel were further investigated. Nevertheless, there are some other researchers trying to implement more sharp schemes to capture the jump conditions across the interface. In particular, the Ghost Fluid method (GFM, hereafter) as introduced for solving the Poisson equation is also used to capture the velocity jump condition during phase change, as shown by Son and Dhir [20], and the group of Tanguy [21–23], the numerical results are proved to be in good agreement with the real physics, however, as far as the authors know, most of the sharp mass transfer schemes are implemented within the level-set method. Moreover with a very different approach, Schlottke and Welgand [24] indicate that by using a mass averaged density together with the corrected velocities at the interface, the spurious oscillations of the pressure in vicinity of the interface would be greatly suppressed, and a similar approach is further adopted by Ma and Bothe [25]. Nevertheless, there are also some other studies [26–30] not mentioning how to deal with the singular problem of the mass source, and no flow instabilities are reported either. For the other difficulty, it comes from the internal boundary condition of the temperature at the interface. If there is no extreme high heat flux during phase change that the liquid–vapor interface is assumed to keep at the saturation temperature as T_{sat}, it could be viewed as a variant type of internal Dirichlet boundary condition. Tsui et al. [31,29] use an implicit interpolation scheme to calculate the temperature field by taking the internal boundary for temperature into consideration. Sato and Ničeno [28] employ a finite difference scheme for the discretization of the temperature by setting T_{sat} at the interface as the boundary condition. Moreover, as an effective approach, GFM is again used to impose the saturated temperature at the interface [32,22]. However, Esmaeeli and Tryggvason [26,27] show that by adding a heat source term representing the latent heat in the energy equation in vicinity of the mixed cells, the calculation of the temperature field is also correct without imposing a boundary condition for it.

Based on the VOF framework, the present study uses a smooth approach to smear the point mass source to the ambient cells close to the interface, for purpose of preserving the numerical stability. Besides, we will show that by controlling the smearing thickness of δ, the scheme is accurate and robust. On the other hand by using a ghost-cell approach, the saturating temperature is imposed as a internal boundary condition at the liquid–vapor interface during solving the energy equation. At the opposite, we provide numerical evidence that without imposing such an accurate boundary condition for the temperature field, it fails to determine accurately the interface advancing or retreating, and thereby, the whole computations are not reliable.

In addition, as the volume averaged scheme on basis of the VOF method smears the discontinuity of the physical properties across the interface, especially when the electric conductivities vary very much between the two phases, the corresponding errors in solving the electromagnetic field need to be further investigated. Based on the Cartesian grids, a series of sharp interface methods have been proposed to enforce the sharp jump conditions on the boundary during solving the Poisson type equations, among which the most known ones are the ghost-fluid methods and embedded boundary methods (EBM, hereafter) [33,34], which is also used in the phase change flows for tackling such discontinuous problem. Recently in simulating the electrohydrodynamics multiphase flows, some researchers [35,36] have also employed the GFM to keep