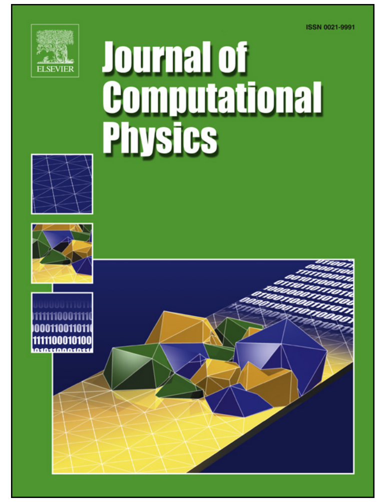
Accepted Manuscript



Direct Numerical Simulation of Incompressible Multiphase Flow with Phase Change

Moon Soo Lee, Amir Riaz, Vikrant Aute

PII: S0021-9991(17)30361-3
DOI: http://dx.doi.org/10.1016/j.jcp.2017.04.073
Reference: YJCPH 7339

To appear in: *Journal of Computational Physics*

Received date: 15 August 2016
Revised date: 15 March 2017
Accepted date: 28 April 2017

Please cite this article in press as: M.S. Lee et al., Direct Numerical Simulation of Incompressible Multiphase Flow with Phase Change, *J. Comput. Phys.* (2017), http://dx.doi.org/10.1016/j.jcp.2017.04.073

Direct Numerical Simulation of Incompressible Multiphase Flow with Phase Change

Moon Soo Lee[a], Amir Riaz[a,*], Vikrant Aute[a]

[a]*Department of Mechanical Engineering, University of Maryland, College Park, MD 20742, USA*

Abstract

Simulation of multiphase flow with phase change is challenging because of the potential for unphysical pressure oscillations, spurious velocity fields and mass flux errors across the interface. The resulting numerical errors may become critical when large density contrasts are present. To address these issues, we present a new approach for multiphase flow with phase change that features, (i) a smooth distribution of sharp velocity jumps and mass flux within a narrow region surrounding the interface, (ii) improved mass flux projection from the implicit interface onto the uniform Cartesian grid and (iii) post-advection velocity correction step to ensure accurate velocity divergence in interfacial cells. These new features are implemented in combination with a sharp treatment of the jumps in pressure and temperature gradient. A series of 1-D, 2-D, axisymmetric and 3-D problems are solved to verify the improvements afforded by the new approach. Axisymmetric film boiling results are also presented, which show good qualitative agreement with heat transfer correlations as well as experimental observations of bubble shapes.

Keywords: Multiphase flow, Phase change, Level set method, Velocity jump conditions

1. Introduction

The thermodynamic state of a fluid dynamic system is altered in response to the transport of thermal energy in a wide range of practical applications including air-conditioning, refrigeration, energy storage and thermal management of electronic devices. The associated physical phenomenon of phase change is complex and cannot always be characterized fully using experimental techniques alone. Advances in computational algorithms and computing power have therefore motivated the study of phase change problems using direct numerical simulation.

Some of the earlier simulations of phase change problems involving deforming interfaces were performed using Lagrangian type methods [1, 2, 3, 4, 5]. More recently, Eulerian type methods for interface capturing have been used effectively in dealing with singular topological events such as the breakup and merging of the interface. Within the Eulerian framework, the interface is typically defined implicitly on a Cartesian grid using either the level set or the volume fraction approach. Such methods have to address the challenge of implementing mass flux and resolving jumps in pressure, velocity, shear stress and temperature gradient across implicit interfaces of arbitrary shapes.

The discretization of interfacial jumps on uniform Cartesian grids can be carried out using either a continuum or a discrete approach. The singular effect of surface tension, for example, can be represented as a distributed volumetric effect on the basis of the continuous surface force (CSF) formulation [6]. A compact support for CSF around the interface can be applied with the help of either the volume of fluid (VOF) [7] or the Dirac delta-function (DDF) [8] method. Alternatively, surface tension effect can be modeled in a discrete manner using the ghost-fluid approach [9]. In the case of phase change, the flux of mass across the interface is related to additional jumps in velocity, momentum and temperature gradient. These effects are often also resolved using either the continuum or the discrete approach.

Within the VOF framework for interface advection, Welch et al. [10] reported simulations of 2-D film boiling using a point mass-source located inside each interfacial cell as a model for interfacial mass flux. The authors validated their

*Corresponding author

Email addresses: lms0523@umd.edu (Moon Soo Lee), ariaz@umd.edu (Amir Riaz), vikrant@umd.edu (Vikrant Aute)