Applied Mathematics and Computation 339 (2018) 636–650

Contents lists available at ScienceDirect

Applied Mathematics and Computation

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

Moving mesh method for direct numerical simulation of two-phase flow with phase change

E. Gros [a,*], G. Anjos [b], J. Thome [a]

[a] Laboratory of Heat and Mass Transfer (LTCM) - École Polytechnique Fédérale de Lausanne (EPFL), Switzerland
[b] Group for Studies and Environmental Simulations in Reservoirs (GESAR) - State University of Rio de Janeiro (UERJ), Brazil

ARTICLE INFO

Keywords:
Finite element method (FEM)
Phase change
Axisymmetric
Two phase flow
Arbitrary Lagrangian–Eulerian (ALE)

ABSTRACT

A moving mesh approach is employed to simulate two-phase flow with phase change. The mathematical model is based on the Arbitrary Lagrangian–Eulerian (ALE) description of the axisymmetric Navier–Stokes equations and energy conservation. These equations are discretized by the Finite Element Method (FEM) on a triangular unstructured mesh in which the phase boundary is represented by a set of interconnected nodes and segments that are part of the computational mesh. Here, phase change and surface tension are implemented as source terms, using the one fluid approach. The method is shown to provide an accurate description of the interfacial forces, heat and mass transfer between phases. Several different verifications are presented where the results are compared to analytical and semi-analytical solutions.

1. Introduction

Two-fluid flow with phase change is at the core of many industrial applications. Some examples are refrigeration systems, nuclear reactors and next generation electronics cooling. Given that the latent heat of vaporization is in general much higher then the thermal capacity, superior heat transfer capabilities are obtained with respect to systems without phase change.

Thanks to the fast increase in computational power, numerical simulations of the coupled Navier–Stokes and heat transfer problem, including phase change, have been possible since about two decades. Here we briefly review the state of the art but the reader is referred to [30] for a recent comprehensive review. Phase change introduces further issues for numerical algorithms like the normal velocity discontinuity, the accurate evaluation of the phase change mass flow rate and sharp boundary layers of the temperature field close to the interface. Early moving mesh finite element methods have been applied to the Stefan problem, solving only for the temperature and interface position. In [7] two-dimensional Stefan problems were solved with space-time finite elements and in [32] freezing around a pipe was simulated. However, it appears that the numerical simulation of the Navier–Stokes coupled to the energy equation with phase change was pioneered by the work of Welch [47]. Welch used a moving unstructured mesh with double degrees of freedom at the interface to simulate evaporation in a closed environment, thermocapillary motion, an oscillating bubble and nucleate boiling. In a second paper [48] Welch, simulated the growth of a vapor bubble on a surface with a pinned contact line. In all these early works, no remeshing was used thus limiting their range applicability to cases without strong interface deformations.

* Corresponding author.
E-mail address: erik.gros@epfl.ch (E. Gros).

https://doi.org/10.1016/j.amc.2018.07.052
0096-3003/

As for the surface tension force, the phase change mass transfer can be included in the interface jump condition or as a singular source term acting at the interface. The SIM method with cut cells (SIMCC) is based on the former approach and it has been used in [18], with a quadratic curve fitting algorithm for the curvature, to simulate the cryogenic flow of a bubble in a tube with phase change. Most computational codes for phase change use the source term approach, with many authors using the VOF method [24,38,49]. Son and Dhir [41] modified the level-set method to include liquid-vapor phase change and investigated film boiling. A coupled level-set and VOF method has been used by Tomar et Al [45] to simulate film boiling and investigate the frequency of bubble formation. The Front Tracking (FT) method has been adapted to phase change problems and applied to simulate phase change by Tryggvason and coworkers [21,29,46]. The FT method has been used to simulate the Stefan problem [20,21], 3D film boiling of a liquid sheet [21], boiling around a cylinder [46], boiling around multiple cylinders [22] and explosive boiling in microgravity [20]. Other examples are [18] where an Eulerian method with cut cells (SIMCC), which is similar to FT, was used to simulate the cryogenic flow of a bubble in a tube with phase change. Sato et al. [37] developed Cartesian grid method, which was used to simulate rising bubbles with evaporation and nucleate boiling.

While most popular methods are based on this macroscopic depiction, there is now increasing interest in computational methods at the mesoscale, where fluid matter is considered a collection of atoms, which is much smaller than macroscale but larger than single atom. Phase field methods offer the possibility to do so, see [1] for a review. The phase-field method was used in [27] to simulate an evaporating bubble in a microchannel, investigating the effects of the contact angle and the results were compared to experimental data. The lattice Boltzmann method (LBM), which is a computational method that solves the Boltzmann equation of statistical mechanics has also been applied to phase change simulations. The LBM was used in [13] to investigated the bubble nucleation and departure within a microchannel.

Moving mesh methods are useful for problems displaying moving boundaries/layers, scale invariance or self similarity [4]. However, moving mesh methods introduce additional degrees of freedom (DOF) to specify the mesh velocities. The mesh velocities can be generated mechanical analogies. A popular approach is to use a mesh which deforms like a fictitious elastic body [12,23,28,43] subjected to displacement boundary conditions at the interface and domain boundary. Several mathematically motivated approaches also exist [4], which try to optimise geometric qualities of the mesh by minimizing a functional or find the mesh velocities as a solution to an additional differential equation.

An important question, in the framework of moving mesh schemes, is the role of the Geometric Conservation Law (GCL), which is discussed in [44]. The GCL originated from the finite volume framework, where it provides a condition to maintain the inherent conservation properties of the FVM. The GCL is usually derived from the requirement [44] that starting from a uniform flow, the numerical scheme must be able to reproduce this uniform flow exactly at all times on a moving mesh. Cao et al. [15] used the GCL to derive a mesh movement algorithm. They impose the Jacobian of the moving domain transformation based on a monitor function and derive a differential equation for the mesh velocity field, which is then shown to be equivalent to the minimization of a functional. A similar method was proposed by Baines et al. [6] based on a local conservation principle to find the mesh velocities. In the so called conservation method, the nodes of the mesh are moved such that the mass fraction associated with particular subregions is preserved. A detailed description of the conservation method is given in [4], along with its application to diverse moving boundary problems. The conservation method has been used to simulate glacier movement [34] and tumour growth in [31], where it was shown to have higher order of convergence than transforming the problem to a fixed domain. In [5] single and two-phase Stefan problems were simulated using only the energy equation (without coupling to the Navier–Stokes equations) and it was shown to converge towards the analytic solution with second order accuracy. In the aforementioned papers, the conservation method was used without changing the number of mesh nodes or their connectivity. Therefore, the mesh tangled when the domain occupied by one phase became to short.

The finite element method (FEM) has shown to be well suited for moving mesh methods for the same reasons which recommend it for fixed-mesh applications [32]: flexibility for irregular geometry arising from nodes motion is easily accomodated and representation of variable coefficients is possible. The extended finite element method (XFEM), which introduces additional (so called “enriched”) interpolation functions on elements cut by the interface, allows it to represent discontinuities of the flow variables with a fixed mesh. In [33] an XFEM with a level-set are implemented and used to simulate solidification problems. The Arbitrary Lagrangian–Eulerian (ALE) approach allows to choose the mesh movement freely by specifying the mesh points velocities. As a result the interface mesh can be moved in a physical way describing the two-phase flow while the remaining mesh nodes may be adjusted with the goal of preserving well shaped mesh elements during the simulation. Severe mesh deformations can occur with moving mesh methods and in the worst case, some mesh elements might even be deformed to a point that prevents the simulation to proceed. Therefore, remeshing can help to handle cases where the motion of the interface would otherwise lead to strong mesh distortions. Remeshing can mean changing only the connectivity of the mesh and/or inserting/removing mesh points. After remeshing, an interpolation is required to project the solution from the old onto the new mesh. This interpolation introduces additional errors, which are usually of diffusive nature, i.e. they tend to smear out steep gradients. In [26] evaporating bubbles in a microchannel are simulated in an ALE framework with the commercial software COMSOL multiphysics and it is said that an automatic remeshing approach is the key issue.

A fundamental question in phase change computations is how to model the mass transfer rate at the interface. Two main categories of models can be distinguished: those based on thermodynamic equilibrium, where the mass transfer rate is computed from the energy jump condition [18,21,37,41,47–49] and models accounting for a departure from