A balanced-force control volume finite element method for interfacial flows with surface tension using adaptive anisotropic unstructured meshes

Zhihua Xie [a,b,*], Dimitrios Pavlidis [a], Pablo Salinas [a], James R. Percival [a], Christopher C. Pain [a], Omar K. Matar [b]

[a] *Applied Modelling and Computation Group, Department of Earth Science and Engineering, Imperial College London, London, UK*
[b] *Department of Chemical Engineering, Imperial College London, London, UK*

ABSTRACT

A balanced-force control volume finite element method is presented for three-dimensional interfacial flows with surface tension on adaptive anisotropic unstructured meshes. A new balanced-force algorithm for the continuum surface tension model on unstructured meshes is proposed within an interface capturing framework based on the volume of fluid method, which ensures that the surface tension force and the resulting pressure gradient are exactly balanced. Two approaches are developed for accurate curvature approximation based on the volume fraction on unstructured meshes. The numerical framework also features an anisotropic adaptive mesh algorithm, which can modify unstructured meshes to better represent the underlying physics of interfacial problems and reduce computational effort without sacrificing accuracy. The numerical framework is validated with several benchmark problems for interface advection, surface tension test for equilibrium droplet, and dynamic fluid flow problems (fluid films, bubbles and droplets) in two and three dimensions.

1. Introduction

Interfacial flows with surface tension appear in many engineering applications, e.g. micro-fluidics, oil-and-gas transportation systems, geophysical flows and nuclear reactors. These applications typically involve the motion of bubbles, droplets, fluid films and jets, featuring tremendous complexity in interfacial topology and a large range of spatial scales.

A key requirement for modelling interfacial flows is a method for tracking or capturing the interface [1]. Numerous methods have been proposed and used to simulate interfacial flows on a fixed mesh, such as marker-and-cell [2], volume-of-fluid (VOF) [1,3,4], front-tracking [5], level set [6,7], phase field [8] and particle [9] methods. In particular, VOF methods are widely used due to the inherent properties of: mass conservation, computational efficiency and easy implementation. From a general point of view, there are two classes of algorithms to solve the transport equation of volume fraction: geometric and algebraic computation [4].

In the geometric VOF methods [1], interfaces are first reconstructed from the volume fraction data so that a geometric profile is found which approximates the actual interface location. Then changes in volume fraction are calculated by integrating volume fluxes across cell boundaries, using flux splitting or unsplitting schemes. In the algebraic computation [10,11], the interface is captured by solving the transport equation of volume fraction with a differencing scheme without reconstructing the interface, such as the flux-corrected transport scheme [10] and using the normalised variable diagram (NVD) [12] concept to switch between different differencing schemes [11].

As the dynamics of interfacial flows are highly unsteady and the shape and location of the interface are changing during the simulation, interface calculation methods based on a fixed mesh need finer mesh resolution in order to capture the details, which will significantly increase computational efforts. The alternative is to consider the use of dynamically adaptive mesh methods, where the mesh resolution can vary in time in response to the evolving solution fields. For example, a finer mesh could be placed around the interface during its development while a coarser mesh could be used away from the interface while the flow is less dynamic. There are some examples of the use of adaptive mesh refinement for structured meshes with volume of fluid [13] and

* Corresponding author.
  E-mail address: zhihua.xie@hotmail.com, z.xie@imperial.ac.uk (Z. Xie).
  URL: http://amcg.ese.imperial.ac.uk (Z. Xie, D. Pavlidis, P. Salinas, J.R. Percival, C.C. Pain)

hybrid level set/front tracking [14] methods. Unstructured meshes are very attractive to deal with complex geometries in engineering applications and there is an example of adaptive unstructured meshes with the level set method [15]. Recently, a novel algebraic VOF interface capturing method based on a compressive advection method on adaptive unstructured meshes has been developed by [16] and some examples of its application for multiphase flows in two dimensions can be found in [17,18]. It is also worth mentioning that many computational domains have a high aspect ratio and most interfacial flow phenomena can possess strong anisotropies, therefore anisotropic mesh resolution may be required to optimally represent the dynamics of the flow. Some examples of anisotropic unstructured mesh adaptivity can be found in [19].

In this paper, we focus on the surface tension force model in three dimensions. Many different types of surface tension force model have been proposed in the past, where the continuum surface force (CSF) method [20] has been widely used in the level set and volume of fluid methods. The level set function is a smooth continuous function, which can estimate accurately the curvature for the surface tension, however the standard level set method might suffer from the mass conservation. The volume of fluid method is mass conservative, however it is difficult to calculate the surface tension force accurately due to the step function of the volume fraction. This leads to the development of coupled level set and volume of fluid (CLSVOF) method [21], which takes advantage of both methodologies. Recently the balanced-force algorithm for surface tension model has become popular in structured Cartesian grids due to the use of a height function for curvature calculation in the volume of fluid method [22] and the level set function in the CLSVOF method [23]. It has also been extended for adaptive mesh refinement for structured Cartesian grids [13]. However, less attention has been paid to the balanced-force algorithm for fully unstructured meshes, even without mesh adaptivity.

The motivation for this work is to develop a balanced-force control volume finite element method for three-dimensional interfacial flows with surface tension on adaptive anisotropic unstructured meshes, which can modify unstructured meshes to better represent the underlying physics of interfacial problems and reduce computational efforts without sacrificing accuracy. A new balanced-force algorithm for the CSF model is proposed within the interface capturing framework based on the volume of fluid method.

The remainder of this paper is organised as follows. Description of the model and numerical methods is given in Section 2. Numerical examples of pure advection, static drop in equilibrium, fluid films, bubbles and droplets are presented in Section 3. Finally, some concluding remarks and future work are given in Section 4.

2. Mathematical model and numerical methods

In this section, we first describe the mathematical model and then we present our numerical framework based on the control volume and finite element method. The new balanced-force algorithm for the CSF model is proposed within the interface capturing framework and discussed in detail.

2.1. Governing equations

In multi-component flows, a number of components exist in one or more phases (one phase is assumed here but is easily generalised to an arbitrary number of phases or fluids). Let α_i be the mass fraction of component i, where $i = 1, 2, .., N_c$ and N_c denotes the number of components. The density and dynamic viscosity of component i are ρ_i and μ_i, respectively. A constraint on the system is:

$$\sum_{i=1}^{N_c} \alpha_i = 1.$$

(1)

For each fluid component i, the conservation of mass may be defined as,

$$\frac{\partial}{\partial t}(\alpha_i) + \nabla \cdot (\alpha_i \mathbf{u}) = 0, \quad i = 1, 2, \ldots, N_c,$$

(2)

and the equations of motion of an incompressible fluid may be written as:

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot [\mu(\nabla \mathbf{u} + \nabla^T \mathbf{u})] + \rho \mathbf{g} + \mathbf{F}_\sigma,$$

(3)

where t is the time, $\mathbf{u}$ is velocity vector, p is the pressure, the bulk density is $\rho = \sum_{i=1}^{N_c} \alpha_i \rho_i$, the bulk dynamic viscosity is $\mu = \sum_{i=1}^{N_c} \alpha_i \mu_i$, $\mathbf{g}$ is the gravitational acceleration vector, and $\mathbf{F}_\sigma$ is the surface tension force. In the present study, we focus on the surface tension model for interfacial flows with two components, i.e. $N_c = 2$.

2.2. Numerical methods

There are several numerical discretisation methods that solve the governing equations, such as the finite difference method, finite volume method and finite element method [24]. The finite element method with unstructured meshes is very attractive, as it provides accuracy and great flexibility in dealing with complex geometries and moving interfaces. In addition, with the finite element method it is possible to develop a compact high-order discretisation by applying higher-order polynomial expansions within every element.

2.2.1. Computational grid

The numerical framework consists of control volume and finite element formulation and also a discontinuous/continuous finite element pair. In the formulation, the domain is discretised into triangular or tetrahedral elements and in this work, they are either P_1DG-P_1 elements (linear discontinuous velocity between elements and linear continuous pressure between elements) or P_1DG-P_2 elements (linear discontinuous velocity between elements and quadratic continuous pressure between elements) [16]. Fig. 1 shows the locations of the degrees of freedom for the P_1DG-P_1 and P_1DG-P_2 elements and the boundaries of the control volumes in two dimension (2D).

2.2.2. Temporal discretisation

Time stepping schemes include first-order schemes, such as the explicit forward Euler and implicit backward Euler schemes. The explicit scheme is more easy and straightforward to implement but imposes restriction on the time step size due to the Courant–Friedrichs–Lewy (CFL) condition, whereas the implicit scheme is stable for large Courant numbers but is more dissipative. A new time discretisation scheme is employed here. When high-order discretisation is sought, the method is based on traditional Crank–Nicolson time stepping. The Crank–Nicolson method is often used because it has the simplicity of a two-level time stepping method, is unconditionally stable and second-order accurate. However, for interface-capturing applications, the time discretisation scheme is based on the explicit forward Euler time stepping method. This introduces negative dissipation and is thus a compressive scheme which helps maintain sharp interfaces. The use of time steps of the order of the grid Courant number and above can result in numerical oscillations and unphysical solutions. For this reason an adaptive θ parameter is introduced [16] and shown explicity in Section 2.2.3, in which the forward Euler time stepping method is obtained for $\theta = 0$, the Crank–Nicolson method is obtained for $\theta = 0.5$ and the backward Euler method is obtained for $\theta = 1$.