Chemical Engineering Journal 349 (2018) 662–682

Contents lists available at ScienceDirect

Chemical Engineering Journal

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

Numerical approach to study bubbles and drops evolving through complex geometries by using a level set – Moving mesh – Immersed boundary method

E. Gutiérrez[a,b], F. Favre[c], N. Balcázar[a,b], A. Amani[a], J. Rigola[a,*]

[a] Heat and Mass Transfer Technological Center (CTTC), Universitat Politècnica de Catalunya – BarcelonaTech (UPC) ESEIAAT, Colom 11, 08222 Terrassa (Barcelona), Spain

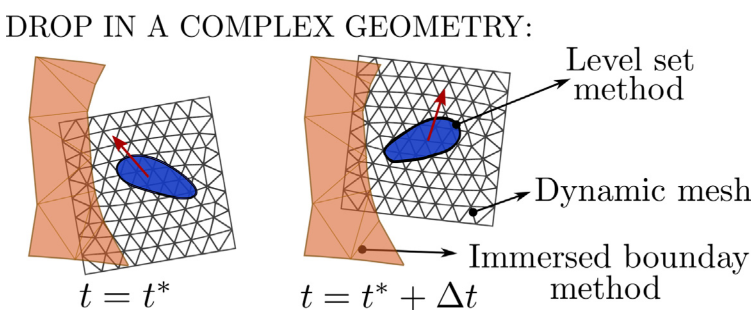
[b] Termo Fluids S.L., Avda Jacquard 97 1-E, 08222 Terrassa (Barcelona), Spain

[c] Instituto de Ingeniería Mecánica y Producción Industrial (IIMPI), Universidad de la República, Uruguay

HIGHLIGHTS

- A new formulation is proposed to study drops/bubbles in complex geometries.
- The multiphase domain is successfully tackled using a conservative level set method.
- The simulation domain is optimized by using a moving mesh.
- Inner and intricate boundaries are handled by using an immersed boundary method.
- Extensive numerical tests were conducted in order to validate the proposed method.

GRAPHICAL ABSTRACT



ARTICLE INFO

Keywords:
Complex geometries
Arbitrary Lagrangian-Eulerian formulation
Level set method
Immersed boundary method
Multiphase flow
Unstructured meshes

ABSTRACT

The present work proposes a method to study problems of drops and bubbles evolving in complex geometries. First, a conservative level set (CLS) method is enforced to handle the multiphase domain while keeping the mass conservation under control. An Arbitrary Lagrangian-Eulerian (ALE) formulation is proposed to optimize the simulation domain. Thus, a moving mesh (MM) will follow the motion of the bubble, allowing the reduction of the computational domain size and the improvement of the mesh quality. This has a direct impact on the computational resources consumption which is notably reduced. Finally, the use of an Immersed Boundary (IB) method allows to deal with intricate geometries and to reproduce internal boundaries within an ALE framework. The resulting method is capable of dealing with full unstructured meshes. Different problems have been studied to assert the proposed formulation, both involving constricting and non-constricting geometries. In particular, the following problems have been addressed: a 2D gravity-driven bubble interacting with a highly-inclined plane, a 2D gravity-driven Taylor bubble turning into a curved channel, the 3D passage of a drop through a periodically constricted channel, and the impingement of a 3D drop on a flat plate. Good agreement was found for all these cases study, which proves the suitability of the proposed CLS + MM + IB method to study this type of problems.

* Corresponding author.
E-mail addresses: kike@cttc.upc.edu (E. Gutiérrez), quim@cttc.upc.edu (J. Rigola).

https://doi.org/10.1016/j.cej.2018.05.110
Received 14 February 2018; Received in revised form 18 May 2018; Accepted 19 May 2018

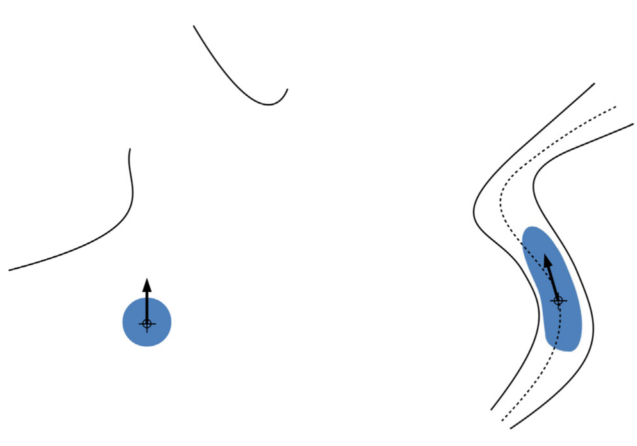
1. Introduction

The motion of drops and bubbles in complex geometries is of fundamental importance in many scientific and engineering applications. To cite a few examples, chemical reactors generally involve many drop-wall collision processes [1], and its understanding could seriously determine the efficiency of the reactor. The field of microfluidics [2] and lab-on-a-chip concept are fed from the knowledge of the behaviour of bubbles and drops evolving through microgeometries. Additionally, the oil extraction processes could ultimately be reduced to the evolution of a slug flow through constricting solids.

The motion of bubbles and drops in unbound mediums has attracted significant scientific attention in the last decades (see Tryggvason et al. [3] for an extensive numerical review). On the contrary, the literature about bubbles/drops evolving through complex geometries is far more limited. An meaningful distinction within these problems is stressed here, depending on the relation between the secondary phase and the surrounding geometry. On the one hand, the solid could constrict the bubble or drop, and its trajectory is somehow predetermined by the own shape of the solid. On the other hand, the bubble/drop could freely evolve in an unbounded media, whereas the present solids alter its motion, but in an unconstrained manner. The border between both types of cases is diffuse, and the classification of a specific problem in one group or the other can be ambiguous. See Fig. 1 for a graphical interpretation of both types of problems.

In order to face both types of problems, different approaches have been proposed in the literature. Experimental procedures usually isolate the basic phenomenon to macroscopically study the motion of the secondary phase. See [4–7] for some valuable experimental works. Additionally, the problem of drops or bubbles evolving in complex geometries can also be addressed theoretically by simplifying the governing equations to extract analytical conclusions (see e.g. [8,9]). Finally, some valuable numerical approaches have been conducted to solve the aforementioned problem [10–13]. Table 1 compiles some of the outstanding works present in the literature, highlighting the method used to solve the problem and the relationship between drop/bubble and geometry.

When facing this type of problems by using a numerical approach, three paramount issues should be addressed in order to satisfactorily solve the case study:



(a) (b)

Fig. 1. Two main scenarios may appear when studying the evolution of bubbles/drops in complex geometries: (a) an unconstrained situation, in which the geometry does not determine beforehand the movement of the bubble/drop; and (b) a constrained situation where a tubular geometry forces the movement of the bubble/drop following a driving curve.

(i) The fluid interface must be computed accurately while conserving integral properties.
(ii) The computational cost should be kept within reasonable bounds.
(iii) The solid geometries, which could be complex and intricate, should be represented effectively and robustly.

Regarding the first item, there are two main groups of methods to deal with multiphase domains. On the one hand, the interface between fluids could be reproduced by using a Front-Tracking method [24]. These techniques accurately describe the multiphase flow, although their implementation may be burdensome due to the need of recomputing the mesh at each time step. On the other hand, the eulerian methods represent the multiphase domain by a continuous (though sharp) change of properties. Those methods include volume-of-fluid (VoF) techniques [25], level set (LS) methods [26,27] and hybrid procedures (CLSVOF) [28]. Level set approaches have the advantage of precisely calculating the geometrical properties of the interface (i.e. normal and curvature). However, they present mass conservation drawbacks. On the contrary, the volume-of-fluid methods inherently conserve mass, but at the expense of a troublesome process of computing geometrical properties of the interface. Hybrid methods solve the two issues present in the above-mentioned techniques, but the computational cost significantly increases. In the present work, we propose a methodology based on a conservative level set (CLS) formulation for unstructured meshes, first reported by Balcázar et al. [29]. The CLS formulation dramatically reduces the mass conservation error in comparison with a standard level set method. This technique has been thoroughly verified [30,31].

Further efforts have been reported in the development of conservative level-set methods, e.g. the level set remedy approach based on sigmoid function [32], and the accurate conservative level-set method [33]. In the present CLS formulation [29], interface normals are computed using a least-squares method on a wide and symmetric nodes-stencil around the vertexes of the current cell [29]. These normals are then used for an accurate computation of surface tension, without additional reconstruction of the distance function, as in geometrical volume-of-fluid/level-set methods [28] or fast-marching methods [33]. Moreover, most computational operations are local. Therefore this method is efficiently implemented for parallel platforms [29,34]. The CLS method has been designed for general unstructured meshes [29]. Indeed, the grid can be adapted to any domain, enabling for an efficient mesh distribution in regions where interface resolution has to be maximized [28,29,31,34,35], which is difficult by using structured grids. Furthermore, a TVD flux-limiter scheme [29] is used to advect the CLS function, avoiding numerical oscillations around discontinuities, whereas the numerical diffusion is minimized. Finally, the present finite-volume formulation is attractive due to its simplicity and the satisfaction of the integral forms of the conservation laws over the entire domain [29].

When facing the problem of a bubble/drop evolving in complex geometries by using DNS methodologies, the computational resources consumption should be a topic of major concern. This is because the need of enough resolution to represent real geometries, together with the high-demanding process of solving the Navier-Stokes equations. With the exception of basic configurations, a decision should be taken regarding this point. An option is to work under a 2D or axisymmetric hypothesis [11,22]. However, if a full 3D approach is sought, a domain optimization method becomes mandatory (e.g. non-inertial reference frame, periodic domain, etc.). In the present work, we enforce a moving mesh (MM) technique to deal with small simulation domains. This Arbitrary Lagrangian-Eulerian (ALE) formulation is based upon the work of Estruch et al. [36]. The mesh follows the motion of the bubble/drop. Under those circumstances, the simulation domain can be limited to the important regions of the problem (i.e. the bubble/drop and its surroundings). This allows a great saving of computational effort,