Journal of Computational Physics 376 (2019) 28–53

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

An efficient method for two-fluid incompressible flows appropriate for the immersed boundary method

C. Frantzis *, D.G.E. Grigoriadis

Computational Sciences Laboratory, UCY-CompSci, Department of Mechanical & Manufacturing Engineering, University of Cyprus, 75 Kallipoleos, Nicosia 1678, Cyprus

ARTICLE INFO

Article history:
Received 3 August 2017
Received in revised form 4 September 2018
Accepted 17 September 2018
Available online 20 September 2018

Keywords:
Two-fluid incompressible flows
Fast direct solvers
Immersed boundary method
Conservative level-set
Interfacial flows
Multiphase flows

ABSTRACT

The numerical simulation of two-fluid flows with sharp interfaces is a challenging field, not only because of their complicated physical mechanisms, but also because of increased computational cost. An efficient and robust numerical formulation for incompressible two-fluid flows is proposed. Its novelty is the consistent coupling of Fast Direct Solvers (FDS) with the Immersed Boundary (IB) method to represent solid boundaries. Such a coupling offers several advantages. First, it extends the range of applicability of the IB method. Second, it allows the simulation of practical problems in geometrically complicated domains at a significantly reduced cost. Third, it can shed light on regions of the parametric space which are considered out of reach, or even impossible today.

Instead of using a conventional variable coefficient pressure Poisson equation, a pressure-correction scheme is suggested for the solution of a constant coefficient Poisson equation for the pressure difference, extending the novel work of Dodd and Ferrante [8]. The conservative Level-set (LS) method is used to track the interface between the two fluids. Appropriate schemes, based on the local directional Ghost Cell Approach (GCA) are proposed, in order to satisfy the boundary conditions (BCs) of the pressure and the LS function around the IB.

The accuracy, robustness, and performance of the proposed method is demonstrated by several validations against conventional approaches and experiments. The results verify that the pressure BCs are properly recovered along the IB solid interface, while a non-smooth pressure field is also allowed across the solid obstacle. The accuracy of the method was found to be 2nd-order, both in time and space. The performance of the proposed method is compared against the conventional approach using a multigrid iterative solver. The impact of the time-step on the accuracy of the constant coefficient approach is examined. Results show that the final speed-up strongly depends on the specific physical and numerical parameters such as the density ratio or the Reynolds number. It is demonstrated that for the range of parameters examined, speed-up factors of 100–10 can be achieved for density ratios of 10–1000 respectively.

* Corresponding author.
E-mail addresses: cfrant02@ucy.ac.cy (C. Frantzis), grigoria@ucy.ac.cy (D.G.E. Grigoriadis).

https://doi.org/10.1016/j.jcp.2018.09.035

1. Introduction

The accurate and efficient simulation of incompressible viscous flows for two immiscible fluids is a challenging task in computational fluid dynamics (CFD). This class of problems appears in various multiphase flows of practical interest which involve interfaces of two liquids or gas/liquids such as wave flows, droplets, bubbles etc. The most widely used technique to solve the Navier–Stokes (NS) equations is the projection method [6], where the calculation of the velocity and pressure is decomposed by solving a Poisson equation for the pressure in order to satisfy the continuity equation. In the projection method, the solution of the Poisson equation is usually the most time-consuming part of the whole procedure.

One of the most robust and efficient ways to solve the pressure Poisson equation, for single-fluid flows with no density variations, is the use of Fast Direct Solvers (FDS) [43]. These solvers require that the associated matrix of coefficients does not change in time.

For variable density or two-fluid flows, the solution of pressure becomes even more computationally demanding than the single-fluid case with constant density. This is a result of the variable coefficients of the derived Poisson equation, which are defined not only by the local grid spacing but also by the local density. Therefore, depending on the motion of the interface, these coefficients should be modified in space and time accordingly, even if the grid is not deforming. Under these conditions, efficient and robust FDS such as FISHPACK [45] are simply not applicable, because they demand constant coefficients for the Poisson equation. Although the computational cost to reassemble the matrix of coefficients is not significant, the use of conventional direct solvers (such as LU-decomposition solvers) is simply computationally prohibited because of the need to invert a huge matrix at each time-step. The most popular solvers are the iterative ones, which perform slower when compared to the FDS, because of the large number of iterations required for convergence, and not due to the additional cost to resemble the matrix of coefficients.

Several attempts have been made to address these challenges. For example, Guermond and Salgado [16] have used a penalty formulation to reduce the computational cost by solving a constant coefficient Poisson equation. Later, Dong and Shen [9] proposed a velocity-correction method to solve the two-fluid NS equations using the phase-field approach, where the Poisson equation for the pressure is transformed from a variable to a constant coefficients one. Recently, Dodd and Ferrante [8] used the idea of Dong and Shen [9] to develop a pressure-correction method to simulate the flow of two incompressible and immiscible fluids with large density and viscosity ratios. Using the Volume of Fluid (VoF) method to track the interface, they demonstrated that the accuracy of their method depends on the type of extrapolation to estimate the pressure in the next time-step. Moreover, they showed that the solution of the pressure Poisson equation can be accelerated as much as 60 times using FDS compared to the multigrid iterative solvers.

The use of a FDS increases the parallel efficiency of the pressure solution, mainly because of the use of Fast Fourier Transformation (FFT) decomposition. This is how a large system of linear equations is transformed into multiple smaller systems that can be solved considerably faster, using a combination of Gaussian elimination and cyclic reduction. On the other hand, this approach has its limitations as well. First, only orthogonal Cartesian or cylindrical grids can be used. In addition, the grid must be uniform along the direction of FFT decomposition. Moreover, a continuous domain of solution is required, i.e. the domain cannot be interrupted by internal obstacles.

FDS have been extensively used for single-fluid problems in combination with the Immersed Boundary (IB) method, after the pioneering work of Peskin [35] to treat problems with moving boundaries. Thereafter, IB has successfully been applied by many researchers to simulate problems with stationary or moving boundaries, allowing simulations in geometrically complicated domains, using efficient FDS on Cartesian grids. However, when it comes to the simulation of two-fluid flows, the use of the IB method to describe the solid boundaries, is not trivial. For example, if one tries to combine the original constant coefficient formulation proposed by Dodd and Ferrante [8] with the IB method, using a FDS, one realises that this is not feasible. This is because the pressure solution is coupled between the fluid and the solid nodes, whereas the NS do not apply in the solid region.

This conflict also appears if a variable coefficient Poisson equation is attempted with the IB method. In order to overcome this issue, a special treatment is required for the nodes close to the IB solid interface. This results to a modified linear system of equations for the pressure field, so that proper boundary conditions (BCs) can be imposed along the IB. For example, Berthelsen and Faltinsen [2] proposed a local directional Ghost Cell Approach (GCA) that is able to manage two-fluid problems in complex geometries; an interesting technique that has also been implemented in open-source CFD codes such as REEF3D [3]. On the other hand, Liu and Lin [26] developed a direct forcing IB approach, called the Virtual Boundary Force (VBF) method, which is also applicable in two-fluid problems. In several previous studies, the IB method has been used to simulate solid obstacles in two-fluid problems, combined with VoF [38,53] or Level-Set (LS) [4,51] methods to track the interface. All of these attempts employ iterative solvers to solve a variable coefficient Poisson equation. This is done in order to locally modify the matrix of coefficients, decoupling the solid phase from the fluid (i.e. excluding the solid nodes from the continuity equation) and satisfy the proper pressure BC along the IB solid interface. Only a few recent studies, and after the innovative work of Dodd and Ferrante [8], examined two-fluid flow problems using a FDS for the solution of the Poisson [12]. However, the authors are not aware of studies that employ FDS and the IB method to describe the solid obstacles in two-fluid flow problems.

The aim of the current work is to extend the constant coefficients approach for two-fluid problems, in order to allow the use of FDS with the IB method to describe complex solid boundaries in a consistent manner, for the first time. The necessary modifications that have to be made to allow the coupling of FDS with IB are reported. The proposed formulation