Journal of Computational Physics 273 (2014) 416–434

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

# A fast pressure-correction method for incompressible two-fluid flows

Michael S. Dodd [1], Antonino Ferrante [*,2]

*William E. Boeing Department of Aeronautics & Astronautics, University of Washington, Seattle, WA 98195, USA*

ARTICLE INFO

*Article history:*
Received 17 October 2013
Received in revised form 7 March 2014
Accepted 19 May 2014
Available online 28 May 2014

*Keywords:*
Variable density Navier–Stokes equations
Projection method
Volume-of-fluid method
Multiphase flow
Interfacial flow
Turbulent flow
Direct numerical simulation

ABSTRACT

We have developed a new pressure-correction method for simulating incompressible two-fluid flows with large density and viscosity ratios. The method's main advantage is that the variable coefficient Poisson equation that arises in solving the incompressible Navier–Stokes equations for two-fluid flows is reduced to a constant coefficient equation, which can be solved with an FFT-based, fast Poisson solver. This reduction is achieved by splitting the variable density pressure gradient term in the governing equations. The validity of this splitting is demonstrated from our numerical tests, and it is explained from a physical viewpoint.

In this paper, the new pressure-correction method is coupled with a mass-conserving volume-of-fluid method to capture the motion of the interface between the two fluids but, in general, it could be coupled with other interface advection methods such as level-set, phase-field, or front-tracking. First, we verified the new pressure-correction method using the capillary wave test-case up to density and viscosity ratios of 10,000. Then, we validated the method by simulating the motion of a falling water droplet in air and comparing the droplet terminal velocity with an experimental value. Next, the method is shown to be second-order accurate in space and time independent of the VoF method, and it conserves mass, momentum, and kinetic energy in the inviscid limit. Also, we show that for solving the two-fluid Navier–Stokes equations, the method is 10–40 times faster than the standard pressure-correction method, which uses multigrid to solve the variable coefficient Poisson equation. Finally, we show that the method is capable of performing fully-resolved direct numerical simulation (DNS) of droplet-laden isotropic turbulence with thousands of droplets using a computational mesh of $1024^3$ points.



## 1. Introduction

In the incompressible flow of two immiscible fluids, the density is discontinuous at the interface, and the density ratio can be of order one thousand (e.g., air–water). Thus, simulating two-fluid flows using a constant density approximation, e.g., the Boussinesq approximation, is not justified. Therefore, the density change between the two fluids must be accounted for directly when solving the governing equations of fluid motion. From a numerical point of view, this introduces additional

* Corresponding author.
*E-mail addresses:* doddms@uw.edu (M.S. Dodd), ferrante@aa.washington.edu (A. Ferrante).

[1] Ph.D. candidate, William E. Boeing Department of Aeronautics & Astronautics, University of Washington, Seattle.
[2] Assistant Professor, William E. Boeing Department of Aeronautics & Astronautics, University of Washington, Seattle.

http://dx.doi.org/10.1016/j.jcp.2014.05.024
0021-9991/

challenges which are not present when solving incompressible single-fluid flows. These challenges are to obtain a numerical method that is computationally efficient, second-order accurate, and stable for large density ratios.

A common solution technique for the incompressible Navier–Stokes equations is the projection method [1]. In the projection method, a Poisson equation for pressure must be solved numerically at each time step. This operation takes most of the computational time in the projection method. Consequently, much work has gone into developing so-called "fast Poisson solvers", which use a combination of fast Fourier transforms (FFT) and Gauss elimination to solve the Poisson equation directly in Fourier space (e.g., [2–5]). However, fast Poisson solvers cannot solve the Poisson equation for pressure that arises in two-fluid flows when advancing the solution from time $t^n$ to $t^{n+1}$ ($\Delta t = t^{n+1} - t^n$):

$$\nabla \cdot \left( \frac{1}{\rho^{n+1}} \nabla p^{n+1} \right) = \frac{1}{\Delta t} \nabla \cdot \mathbf{u}^*, Al \tag{1}$$

where $\rho$ is the density, $p$ is the pressure, and $\mathbf{u}^*$ is the approximate velocity field at $t^{n+1}$. Fast Poisson solvers require a constant coefficient on the left-hand side of the Poisson equation, whereas the coefficient $(1/\rho^{n+1})$ on the left-hand side of Eq. (1) varies in space and time. To solve the variable coefficient equation (1), a standard practice has been to use iterative methods, e.g., multigrid methods [6–8], Krylov methods [9,10] or Krylov methods preconditioned with multigrid [11]. The downside is that iterative methods are often slower than fast Poisson solvers. In fact, they can be up to ten times slower [2]. Additionally, an iterative method's operation count depends on problem parameters (e.g., density ratio) and convergence tolerance, whereas fast Poisson solvers have the advantage of fixed operation counts.

To reduce the computational time in solving variable density incompressible flows, Guermond and Salgado [12] adopted a penalty formulation, whereby only a constant coefficient Poisson equation must be solved at each time step. More recently, for solving the two-fluid coupled Navier–Stokes Cahn–Hilliard (phase-field) equations, Dong and Shen [13] developed a velocity-correction method that transforms the Poisson equation (1) from a variable to a constant coefficient equation. The underlying idea is to split the variable-coefficient pressure-gradient term into a constant term and a variable term, and then treat the constant term implicitly and the variable term explicitly as

$$\frac{1}{\rho^{n+1}} \nabla p^{n+1} \to \frac{1}{\rho_0} \nabla p^{n+1} + \left( \frac{1}{\rho^{n+1}} - \frac{1}{\rho_0} \right) \nabla \hat{p}, \tag{2}$$

where $\rho_0$ is a constant to be defined in Section 2 and $\hat{p}$ is an explicit approximation to the pressure at time level $n+1$, specifically,

$$\hat{p} = \begin{cases} p^n & \text{if } J = 1, \\ p^* = 2p^n - p^{n-1} & \text{if } J = 2, \end{cases} \tag{3}$$

where the choice of $J = 1$ or 2 indicates, respectively, constant or linear extrapolation of $p^{n+1}$.[3] Then, applying the substitution (Eq. (2)) to the left-hand side of Eq. (1), leads to a constant coefficient Poisson equation for pressure,

$$\nabla^2 p^{n+1} = \nabla \cdot \left[ \left( 1 - \frac{\rho_0}{\rho^{n+1}} \right) \nabla \hat{p} \right] + \frac{\rho_0}{\Delta t} \nabla \cdot \mathbf{u}^* \tag{4}$$

that can be solved directly using a fast Poisson solver. Also, in contrast to solving Eq. (1), there is no need to assemble variable coefficient matrices at each time step when solving Eq. (4). Therefore, by using the splitting technique in Eq. (2), there is potential for significant speedup in solving the Navier–Stokes equations for incompressible two-fluid flows.

In this paper, we aim to answer the following questions on applying the splitting technique (Eq. (2)) to the two-fluid Navier–Stokes equations:

- Is the split method (Eq. (4)) physically less accurate than the unsplit method (Eq. (1)) and does the accuracy of the split method depend on the type of extrapolation chosen in Eq. (3)?
- What are the computational savings of using the split method? Specifically, how much faster is it to solve Eq. (4) directly versus Eq. (1) iteratively?

To answer these questions, we first develop a two-fluid pressure-correction method with a constant coefficient Poisson equation, which can be solved directly with a fast Poisson solver. Next, we apply the method to several two-fluid flows to verify and validate the new method, quantify its performance, and evaluate its spatial and temporal accuracy.

The paper is organized as follows: in Section 2 we describe the new pressure-correction method for uniform Cartesian grids and periodic boundary conditions and its coupling to the volume-of-fluid (VoF) method. Section 3 compares the numerical solutions obtained when using the standard, unsplit method (i.e., solving Eq. (1), called Unsplit), the split method, Eq. (2), using $J = 1$ (called FastP$^n$), and the split method using $J = 2$ (called FastP$^*$). Next, we assess the performance of the unsplit and split formulations by comparing the CPU time when solving Eq. (1) using a multigrid solver to the CPU time

[3] A similar strategy for handling variable coefficients in the advection–diffusion equation was proposed by Gottlieb and Orszag [14, Section 9, p. 114].