Journal of Computational Physics 249 (2013) 185–203

Contents lists available at SciVerse ScienceDirect

Journal of Computational Physics

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

# A monotonicity preserving conservative sharp interface flow solver for high density ratio two-phase flows

Vincent Le Chenadec [a,*], Heinz Pitsch [a,b]

[a] Department of Mechanical Engineering, Stanford University, CA 94305, USA
[b] Institute for Combustion Technology, RWTH Aachen, Templergraben 64, 52056 Aachen, Germany

ARTICLE INFO

*Article history:*
Received 16 October 2012
Received in revised form 18 April 2013
Accepted 20 April 2013
Available online 9 May 2013

*Keywords:*
Volume-of-Fluid
Interfacial flow
Two-phase flow
Free surface flow
Multiphase flow
Non-oscillatory
Monotonicity preserving
Momentum
Conservation

ABSTRACT

This paper presents a novel approach for solving the conservative form of the incompressible two-phase Navier–Stokes equations. In order to overcome the numerical instability induced by the potentially large density ratio encountered across the interface, the proposed method includes a Volume-of-Fluid type integration of the convective momentum transport, a monotonicity preserving momentum rescaling, and a consistent and conservative Ghost Fluid projection that includes surface tension effects. The numerical dissipation inherent in the Volume-of-Fluid treatment of the convective transport is localized in the interface vicinity, enabling the use of a kinetic energy conserving discretization away from the singularity. Two- and three-dimensional tests are presented, and the solutions shown to remain accurate at arbitrary density ratios. The proposed method is then successfully used to perform the detailed simulation of a round water jet emerging in quiescent air, therefore suggesting the applicability of the proposed algorithm to the computation of realistic turbulent atomization.



## 1. Introduction

Since the seminal work of Harlow and Welch [1], the development of numerical techniques for Direct Numerical Computation of realistic free surface flows has been an active area of research. As of now, in spite of the improved predicting capabilities of the algorithms proposed over the years, no standard approach has emerged. Indeed, the numerical representation of two immiscible fluids with different thermodynamic properties is a challenging task, especially in the multiscale flows typically encountered in, for instance, industrial applications.

In spite of steadily increasing computer power, computational cost still remains a limiting factor. An efficient use of computational resources is therefore critical to reach flow conditions encountered in energy conversion devices for instance. Efficient use of a given mesh suggests sharp interface methods, which consist of resolving the interface front over a single cell, as a relevant approach. Such treatment of the interface singularity is obviously prone to stability issues, for it may involve density variations of two or more orders of magnitude. While the use of low pass filters [2] and/or artificial viscosity [3] is often required, in the case of under-resolved computations or de-aliasing, for example, excessive use of such techniques in Direct Numerical Simulations can be detrimental to the quality and dependability of the results.

* Corresponding author. Tel.: +1 650 723 2938.
*E-mail address:* vlechena@stanford.edu (V. Le Chenadec).

0021-9991/$ - see front matter © 2013 Elsevier Inc. All rights reserved.
http://dx.doi.org/10.1016/j.jcp.2013.04.027

This work proposes an algorithm for the solution of the conservative form of the two-phase incompressible Navier–Stokes equations. It relies on three key components: a geometric integration of the convective momentum transport, a monotonicity preserving momentum rescaling, and a consistent Ghost Fluid projection that includes surface tension effects. The resulting geometric discretization is shown to preserve monotonicity for the density, momentum and velocity fields while simultaneously maintaining the second order interface representation required to accurately capture capillary effects. In addition, a discretely equivalent finite volume formulation is provided to characterize the small, third order conservation error.

The algorithm is tested with a variety of cases and shown to accurately represent convection dominant flows as well as viscous and capillary effects. The localized numerical dissipation associated with the proposed discretization is shown to be a first order effect. The primary atomization computation of a water jet emerging in a quiescent air environment is also presented. The proposed method, combined with an efficient pressure solver [4], is able stably handle the resulting large density ratio (800) of this challenging configuration, and the computational time is shown to remain reasonable (of the order of 100,000 CPU hours for a 400M point structured mesh).

## 2. Governing equations

The following section focuses on the set of equations solved in the one-fluid formulation adopted throughout this work. The presence of two immiscible fluids gives rise to singularities that must be accounted for, and a good understanding of the governing equations is required to ensure consistent discretization.

The incompressible Navier–Stokes equations are first written in strong form, and a brief description of how they are typically solved in a one-fluid formulation is presented. Directly updating the physical (as opposed to conserved) variables using single phase flow discretizations, however, is known to be prone to numerical momentum transfer between phases, potentially leading to unphysical flow features at moderate and high density ratios [5]. Numerical procedures such as TVD Runge–Kutta time integration [6] and upwind discretization of the convective transport [7] have been used to extend the range of application of these non-conservative formulations. Alternatively, discretizations based on the conservative form of the Navier–Stokes equations have been proposed and shown [8,9] to lead to accurate results for arbitrary density ratios. This therefore motivates the derivation of the integral form of the Navier–Stokes equations for the conserved variables, which is presented in a second part. The resulting integral form of the equations is at the core of the current method, and the link with the proposed discretization is introduced in a third and last part, in which a new strategy to preserve monotonicity for density, momentum, and velocity at the cost of a localized introduction of numerical dissipation and a small conservation error is devised.

### 2.1. Strong form

In the absence of mass transfer between phases, and if both fluids are viscous, the velocity jump across the interface is null. While the thermodynamic properties remain discontinuous, the continuity of the velocity field greatly simplifies the implementation of the momentum equation if the strong non-conservative form of the Navier–Stokes equations is used. This is emphasized in the following section, which introduces the strong forms of the marker and Navier–Stokes equations, and how they are typically solved in the incompressible limit.

An accurate prediction of the interface location is the cornerstone of predictive two-phase flow computations. In front capturing methods, the interface is transported by means of a mapping to an iso-contour of a marker function $f$. The evolution of $f$ is then governed by

$$\frac{\partial f}{\partial t} + \mathbf{u} \cdot \nabla f = 0. \tag{1}$$

The choice of the numerical scheme used to discretize Eq. (1) depends on the nature of $f$, which may be chosen to be a smooth (Level Set) or heaviside (Volume-of-Fluid) function for example. In the proposed work, we adopt a Volume-of-Fluid (VoF) viewpoint, in which the marker function $f$ is the indicator function

$$f(\mathbf{x}, t) = \begin{cases} 0 & \text{if } \mathbf{x} \text{ is in the gas phase at time } t, \\ 1 & \text{if } \mathbf{x} \text{ is in the liquid phase at time } t. \end{cases} \tag{2}$$

In VoF methods, in order to limit the numerical diffusion of the interface front, the solution of Eq. (1) follows from a two-step procedure [10]. First, given a field of cell-averaged indicator function values (volume fraction $F$), a discontinuous reconstruction of the interface is computed. From this representation, the partial differential equation (1) is solved by evaluating the volume fraction fluxes in a geometric (also referred to as sharp) fashion.

In the incompressible limit of the Navier–Stokes equations, the flow field in both gas and liquid phases is governed by

$$\begin{cases} \nabla \cdot \mathbf{u} = 0, \\ \rho\left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u}\right) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{b}, \end{cases} \tag{3}$$