Journal of Computational Physics 350 (2017) 97–116

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Computation of three-dimensional multiphase flow dynamics by Fully-Coupled Immersed Flow (FCIF) solver

Sha Miao, Kelli Hendrickson, Yuming Liu *

Department of Mechanical Engineering, Massachusetts Institute of Technology, Cambridge, MA 02139, USA

ARTICLE INFO

Article history:
Received 18 May 2017
Received in revised form 6 August 2017
Accepted 20 August 2017
Available online xxxx

Keywords:
Immersed boundary (IB)
uRANS
Long-wave model
Slug generation
Two-phase flow
Cartesian grid

ABSTRACT

This work presents a Fully-Coupled Immersed Flow (FCIF) solver for the three-dimensional simulation of fluid–fluid interaction by coupling two distinct flow solvers using an Immersed Boundary (IB) method. The FCIF solver captures dynamic interactions between two fluids with disparate flow properties, while retaining the desirable simplicity of non-boundary-conforming grids. For illustration, we couple an IB-based unsteady Reynolds Averaged Navier Stokes (uRANS) simulator with a depth-integrated (long-wave) solver for the application of slug development with turbulent gas and laminar liquid. We perform a series of validations including turbulent/laminar flows over prescribed wavy boundaries and freely-evolving viscous fluids. These confirm the effectiveness and accuracy of both one-way and two-way coupling in the FCIF solver. Finally, we present a simulation example of the evolution from a stratified turbulent/laminar flow through the initiation of a slug that nearly bridges the channel. The results show both the interfacial wave dynamics excited by the turbulent gas forcing and the influence of the liquid on the gas turbulence. These results demonstrate that the FCIF solver effectively captures the essential physics of gas–liquid interaction and can serve as a useful tool for the mechanistic study of slug generation in two-phase gas/liquid flows in channels and pipes.

1. Introduction

Two-phase problems involving fluids with disparate flow characteristics interacting at deformable interfaces are common in a wide range of fields. For instance, turbulent gas flow over laminar liquid often appears in the production and transportation of oil/gas flows (e.g. [1]). The study of interaction between turbulent air and inviscid water waves is important in the prediction of ocean surface waves in ocean science and engineering (e.g. [2]). In these applications, the liquid dynamics affect the turbulent gas field through the kinematic constraint of the interface shape and the interfacial particle motion. Meanwhile, the turbulent gas input influences the dynamics of the liquid through interfacial stresses. Understanding these detailed interfacial interactions is key to accurately predicting the flow evolution. Numerical tools capable of capturing this two-way coupling are essential for the mechanistic study of such two-phase flow problems.

Numerically solving flows with disparate flow features is challenging. One approach fully couples two boundary-fitted solvers at the interface [3–5]. This can involve mapping each distorted physical domain into a fixed regular computational domain [3,4]. It can also involve tracking the interface and solving the fluids in each time-varying physical domain [5]. These types of methods are exact with clustered boundary-fitted grids near the interface. Additionally, they allow simplifications

* Corresponding author.
E-mail address: yuming@mit.edu (Y. Liu).

http://dx.doi.org/10.1016/j.jcp.2017.08.042
0021-9991/

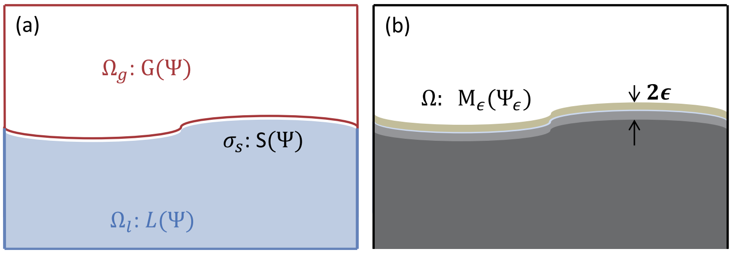

Fig. 1. Conceptual sketch of the two-phase problem. (a) the original two-domain problem with domains Ω_g, Ω_l and their interface σ_s. Their respective governing equations are G, L and S. (b) is the equivalent single-domain problem governed by a meta-equation M_ϵ from the use of the second-order BDIM. A smooth transition between the equations is located within a small distance 2ϵ over the interface σ_s.

on each flow solver based on the physical characteristics of different fluids. However, the implementation is usually complex, requiring grid tracking, re-griding and mapping.

To remove the complexity of boundary-fitted/moving grids, another approach applies a fully immersed single fluid solver on fixed Eulerian grids [6–13]. This approach tracks the interface via methods such as level-set (LS) [10,11], front-tracking [12,13], volume-of-fluid (VOF) [6–8], coupled-level-set-and-volume-of-fluid (CLSVOF) [14], etc. These interface tracking methods are advantageous for flows with complex interfaces such as wave breaking or fingering. However, this approach is limited to one governing equation applied to two different fluids. Computational inefficiencies arise if disparate physical behaviors exist, such as turbulent gas with laminar liquid where the Reynolds numbers in two phases are vastly different [6].

In the present study, we aim at developing a novel three-dimensional Fully Coupled Immersed Flow (FCIF) solver that combines the advantages of these two approaches, i.e., the simplicities of separate fluid solvers *and* the non-moving grids, for the simulation of two-phase flows with disparate flow characteristics. This approach couples different flow solvers on non-boundary-conforming grids by extending an Immersed Boundary (IB) method [15] typically used for fluid–body interaction to non-mixing/breaking fluid–fluid interaction. Within each phase, we select appropriate solvers for the physical characteristics of the fluids. There are many modeling paradigms for IB methods. Good reviews of so-called "diffused interface" schemes via classical IB methods [16–18], direct forcing methods [19,20], and penalization methods [21,22] exist as well as for "sharp interface" schemes such as cut-cell methods [23–25], hybrid Cartesian IB methods [26–28], and curvilinear IB methods [29]. Sharp interface IB methods typically achieve higher precision in implementing interface boundary conditions but are susceptible to non-physical pressure oscillations and have complex implementations for moving boundary problems [30,31]. Diffused interface IB methods are simpler to implement with moving boundaries but suffer the trade off of less rigorous boundary condition implementation. The diffused interface IB method we choose to extend to fluid–fluid interactions in the present work obtains accurate boundary conditions with no pressure oscillations given sufficient resolution.

The application of the FCIF solver considered in this paper is the development of slugs in high-viscosity-oil/high-density-gas flow in horizontal channels and pipes. Experiments have shown that slug flow, a violent intermittent flow pattern commonly observed in multiphase flows, is more probable for heavy oil–gas flow. The typical Reynolds numbers for this flow are $Re_g \sim O(10^3)$–$O(10^5)$ for the turbulent gas and $Re_l \sim O(1)$–$O(10)$ for the laminar liquid [32,33]. The large size and fast speed of the slug bodies provide a number of significant engineering challenges such as structural vibration, pipe erosion/corrosion, reduction of flow transportation efficiency, etc. The physical mechanism behind the slug generation in these heavy-oil/high-density-gas flows remains unclear. To simulate this problem, we apply the FCIF solver with the gas turbulence at high Reynolds number computed by an IB-based uRANS simulator and the liquid dynamics at low Reynolds number computed by a depth-integrated model. In this work, we show that the FCIF solver (with two appropriately chosen fluid solvers) captures the essential physics of turbulent gas-laminar liquid interaction as well as the nonlinear wave evolution for a mechanistic study of the slug generation problem.

The structure of the paper is as follows. §2 describes the extension of the IB for non-mixing/breaking fluid–fluid interaction to achieve the general coupling framework of FCIF solver. §3 provides the development of the gas and liquid solvers, and the details of the coupling framework. §4 presents a series of systematic validations of the FCIF solver. §5 showcases the capability and effectiveness of FCIF for slug development through simulation of a laminar liquid layer sheared by a turbulent gas flow. Finally, §6 draws the conclusions.

2. General FCIF framework

We consider a two-domain interaction problem where one fluid (e.g. gas) occupies domain Ω_g with governing equation $G(\Psi)$, and the other fluid (e.g. liquid) occupies domain Ω_l with governing equation $L(\Psi)$. Ψ represents the field quantities to be solved, such as velocities, turbulent kinetic energy, etc. $S(\Psi)$ satisfies appropriate boundary conditions at the interface σ_s. A conceptual sketch is in Fig. 1(a).

In order to solve $G(\Psi)$ and $L(\Psi)$ using grids that do not conform to the interface σ_s, we apply the IB method of second-order Boundary Data Immersion Method (BDIM) [15]. The second-order BDIM modifies the original two-domain