Journal of Computational Physics 258 (2014) 613–633

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

# A second-order time-accurate implicit finite volume method with exact two-phase Riemann problems for compressible multi-phase fluid and fluid–structure problems

Alex Main [c], Charbel Farhat [a,b,c,*]

[a] Department of Aeronautics and Astronautics, United States
[b] Department of Mechanical Engineering, United States
[c] Institute for Computational and Mathematical Engineering, Stanford University, Stanford, CA 94305-4035, USA

ARTICLE INFO

*Article history:*
Received 25 May 2013
Received in revised form 25 October 2013
Accepted 1 November 2013
Available online 11 November 2013

*Keywords:*
Compressible flow
Finite volume method
Fluid–structure interaction
Hydrostatic
Immersed boundary method
Implicit
Implosion
Multi-material
Multi-phase
Two-phase Riemann solver

ABSTRACT

A family of compressible multi-phase fluid and fluid–structure interaction problems for which implicit schemes are preferable over explicit counterparts is identified. Using as a backdrop a finite volume method based on exact two-phase Riemann problems that has proven to be robust for multi-phase flows with strong contact discontinuities and highly nonlinear fluid–structure interaction problems, an implicit computational framework for the solution of such problems is presented. General issues that arise in the context of second- and higher-order time-discretizations of multi-material problems by multi-step schemes are highlighted, and solutions to these issues are presented in the form of redesigned implicit time-integrators. The proposed implicit computational framework is illustrated with the solution of an air–water shock tube problem, a realistic compressible multi-phase fluid problem, and a highly nonlinear fluid–structure interaction problem associated with the underwater implosion of a cylindrical shell. In all cases, the accuracy and robustness of the proposed implicit computational framework are demonstrated, and its superior computational performance is highlighted.



## 1. Introduction

The simulation of compressible flows involving multiple species of fluids which furthermore may interact with flexible solids or structures are important in many areas, including underwater explosions and implosions, bubble dynamics, detonations, and blasts. These flows are dealt with in the literature under three different names — multi-fluid, multi-material, and multi-phase problems. Multi-fluid and multi-material flows are characterized by the presence of two or more distinct fluids (with different material properties) in well defined regions of space. On the other hand, multi-phase flows may feature different mixtures of fluids (or fluid phases) [1] — for example, a water mist in air. For the purposes of this paper, multi-material, multi-fluid, and multi-phase flows are referred to as "multi-phase", as this is the most general description of such problems.

Simulations of compressible multi-phase flows have been typically performed using either a Lagrangian or Eulerian computational framework. In the Lagrangian framework, the material interface is tracked explicitly, and separate meshes are used for the different fluid subsystems. In the Eulerian framework, a single mesh is typically used and fixed in time; however, in addition to solving for the fluid variables, one may have to solve for the mass fractions (or equivalent quantities) as well. Lagrangian approaches are limited by the necessity to update the meshes at every time-step. On the other hand,

* Corresponding author.
*E-mail addresses:* alexmain@stanford.edu (A. Main), cfarhat@stanford.edu (C. Farhat).

0021-9991/$ – see front matter 
http://dx.doi.org/10.1016/j.jcp.2013.11.001

early attempts at simulating multi-phase flows with Eulerian schemes suffered from oscillations at the material interface [2] and were typically limited to ideal gasses.

The advent of the Ghost Fluid Method (GFM) [3] represented a major step forward. The GFM combines the fixed mesh of an Eulerian scheme with explicit interface tracking using a level set technique. The GFM avoids numerical oscillations at the material interface at the cost of being a non-conservative method. Furthermore, unlike many other alternatives, the GFM allows for the use of arbitrary Equations Of State (EOS). Unfortunately, the GFM is also computationally expensive due to the need to store many "ghost cells". The Ghost Fluid Method for the Poor (GFMP) was introduced in [4] as a computationally lighter alternative to the GFM. However, the GFMP is limited to the stiffened gas EOS. Also, both the GFM and GFMP suffer from numerical instability when the problem of interest involves large contact discontinuities such as those encountered in air–water flows, where the typical density ratio at the material interface is of the order of 1000. These numerical difficulties are completely avoided, however, by the recently developed Finite Volume method with Exact Riemann solvers (FIVER) [5] and its earlier version known as the GFMP with Exact Riemann Solvers [6]. These methods achieve robustness with respect to large density jumps by relying on the solution of exact two-phase Riemann problems at the material interface. Specifically, if one of two neighboring cells $i$ and $j$ is traversed by the material interface, FIVER substitutes the computation of the numerical fluxes $\Phi_{ij}$ at the interface between two neighboring cells $\Omega_i$ and $\Omega_j$ by the computation of the numerical flux based on the fluid state at cell $\Omega_i$ and the solution of an exact two-phase Riemann problem at the material interface on the side of cell $\Omega_i$. Similarly, FIVER substitutes the computation of the numerical flux $\Phi_{ji}$ at the interface between $\Omega_j$ and $\Omega_i$ by the computation of the numerical flux based on the fluid state at cell $\Omega_j$ and the solution of an exact two-phase Riemann problem at the material interface on the side of $\Omega_j$. Like the GFM and GFMP, FIVER is non-conservative at the material interface. However, unlike both of these methods, FIVER is particularly robust with respect to strong contact discontinuities at this interface. Furthermore, it allows for arbitrary EOS by using, when needed, sparse grid tabulations [5]. The effectiveness of FIVER for compressible multi-phase fluid and/or highly nonlinear Fluid–Structure Interaction (FSI) problems has been demonstrated for challenging applications pertaining to underwater implosion and explosion problems [6–8,5].

Popular solution algorithms for coupled FSI problems also fall into two categories: arbitrary Lagrangian–Eulerian (ALE), and Embedded Boundary (EB) methods. ALE methods (for example, see [11]), deform the fluid mesh so that its boundaries conform with the wetted surface of the structure. Unfortunately, such methods struggle when the solid or structural subsystem — referred to throughout the remainder of this paper as the body — undergoes large deformations. This is because the fluid mesh can become significantly warped and eventually undergo mesh crossovers that invalidate it. In the alternative EB methods (for example, see [9]), that are usually based on a purely Eulerian computational approach for the fluid subsystem, the fluid mesh is maintained fixed while the body moves and deforms. Hence, such methods avoid the issue of mesh crossover, but face a number of other difficulties ranging from computational geometry issues related to the identification of the intersection of an embedded surface and an Eulerian fluid mesh [8], to the enforcement of the appropriate wall boundary conditions [7], and the preservation of the desired order of accuracy in the vicinity of the material interface [10]. Nevertheless, unlike mesh crossovers which break ALE simulations, these issues are not in general fatal. In particular, in the presence of a body in a multi-material problem of interest, FIVER also acts as an EB method for Computational Fluid Dynamics (CFD) [7] that distinguishes itself from other EB methods by its equal applicability to structured and unstructured meshes.

Whether in the context of the ALE or EB methods, explicit time-integration has dominated the computational scene for compressible multi-phase fluid and FSI problems. One frequent justification is that these problems are not stability-bound but time-accuracy-bound — that is, their physical time-scale is so small that the computational time-step required for resolving it is of the order of the stability time-step of an explicit time-integration scheme or less, and therefore makes implicit time-integration computationally inefficient. Another frequent argument in favor of explicit time-integration is that even when the physical time-scale is not an issue, multi-phase fluid and FSI problems are so highly nonlinear that the robustness of Newton-like solution methods characterizing implicit schemes becomes an issue. However, three objections are noted here: (1) the robustness and performance of implicit computational technologies have dramatically improved during the last two decades, and (2) there exists a large family of highly nonlinear compressible multi-phase fluid and FSI problems whose solution is not time-accuracy- but stability-bound, and (3) the exclusive reliance in this field on explicit computational technologies leads only to the inefficient solution of such problems, and occasionally makes some of them computationally prohibitive. Indeed, if $u$ denotes for example the velocity of a fluid at an arbitrary point in a one-dimensional compressible multi-phase fluid or FSI problem and $c$ denotes the speed of sound in this fluid, the stability time-step of an explicit scheme designed for the solution of this problem is typically limited by the minimum value of $h/|u+c|$, whereas its accuracy time-step is generally limited by the minimum value of $h/|s|$, where $|s|$ is the speed of the propagating discontinuity and $h$ is the local mesh size. Hence, the time-step is essentially always controlled by the smallest element of the mesh. Despite this, for problems where $|u+c| \gg |s|$, implicit schemes can be expected to be more computationally efficient than their explicit counterparts. Such problems arise in many engineering applications, including those pertaining to hydrostatic implosion. Yet, it seems that little effort has been devoted to the design, analysis, and performance assessment of implicit schemes for such multi-phase fluid and FSI problems. Hence, the main objective of this paper is to fill this gap by describing an implicit extension of FIVER that is second-order time-accurate, and assessing its performance for realistic compressible multi-phase fluid and FSI problems. To this effect, the remainder of this paper is organized as follows.

In Section 2, the compressible multi-phase fluid and FSI problems of interest are formulated. In Section 3, the FIVER method for the solution of such problems is overviewed in order to keep this paper as self-contained as possible. In