Available online at www.sciencedirect.com
ScienceDirect
Journal of Computational Physics 227 (2008) 2083–2104
JOURNAL OF COMPUTATIONAL PHYSICS
www.elsevier.com/locate/jcp

Adaptive solution techniques for simulating underwater explosions and implosions

Samet Y. Kadioglu [a,*,1], Mark Sussman [b,1]

[a] Advanced Nuclear Energy Systems Department, Idaho National Laboratory, P.O. Box 1625, MS 3885, Idaho Falls, ID 83415, United States

[b] Department of Mathematics, 208 Love Building, Florida State University, Tallahassee, FL 32306, United States

Received 18 December 2006; received in revised form 8 August 2007; accepted 16 October 2007
Available online 1 November 2007

Abstract

Adaptive solution techniques are presented for simulating underwater explosions and implosions. The liquid is assumed to be an adiabatic fluid and the solution in the gas is assumed to be uniform in space. The solution in water is integrated in time using a semi-implicit time discretization of the adiabatic Euler equations. Results are presented either using a non-conservative semi-implicit algorithm or a conservative semi-implicit algorithm. A semi-implicit algorithm allows one to compute with relatively large time steps compared to an explicit method. The interface solver is based on the coupled level set and volume-of-fluid method (CLSVOF) [M. Sussman, A second order coupled level set and volume-of-fluid method for computing growth and collapse of vapor bubbles, J. Comput. Phys. 187 (2003) 110–136; M. Sussman, E.G. Puckett, A coupled level set and volume-of-fluid method for computing 3D and axisymmetric incompressible two-phase flows, J. Comput. Phys. 162 (2000) 301–337]. Several underwater explosion and implosion test cases are presented to show the performances of our proposed techniques.

Keywords: AMR; Semi-implicit; CLSVOF; Multi-phase; Under-water explosions and implosions

1. Introduction

In this paper, we present adaptive solution techniques for simulating underwater explosion and implosion problems. These are multi-phase flow problems in which we solve a liquid component (water obeying Tait equation of state) and a gas component (gas is assumed to be spatially uniform, and adiabatic). A multi-phase

* Corresponding author. Tel.: +1 919 259 3648; fax: +208 526 2930.
E-mail addresses: Samet.Kadioglu@inl.gov (S.Y. Kadioglu), sussman@math.fsu.edu (M. Sussman).
1 The authors were supported in part by Office of Naval Research (ONR) STTR under Contract N00014-02-C-0543, and by Weidlinger Associates Inc.

0021-9991/$ - see front matter
doi:10.1016/j.jcp.2007.10.019

flow calculation can be based on either solving a two-fluid system, i.e., both gas and liquid (compressible or incompressible) equations are solved, or based on solving a single fluid system, i.e., only liquid (compressible or incompressible) equations are solved, and the gas contents are treated as spatially uniform. For instance Fedkiw et al. [10,7] used compressible (gas)–compressible (water) and compressible (gas)–incompressible (water) models. Compressible (gas)–compressible (water) models are also used by [23,27,14]. Our method solves the compressible liquid equations together with a compressible, spatially uniform, gas model. Multiphase flow methods, in which the liquid is treated as compressible, are often formulated as an explicit method (i.e. [23,10,27,14]). The time step for an explicit method is constrained by both the magnitude of the underlying velocity field and the magnitude of the sound speed of water. Alternatively, the time step for a semi-implicit method is not constrained by the sound speed of water. The following semi-implicit approaches have been developed for treating water as a compressible fluid [31,30,9,32]. In the context of a multi-phase flow problem, very little attention has been given to semi-implicit approaches for discretizing the equations for the compressible flow of water [33].

Here we use semi-implicit discretizations to solve underwater explosion and implosion problems. The semi-implicit discretization approach enables us to solve many problems involving underwater explosions/implosions with a much larger time step than using an explicit discretization approach. An efficiency comparison between our semi-implicit methods and an explicit method due to Wardlaw [27] is made in the results section.

Adaptive Mesh Refinement (AMR) is another core feature of our algorithms. Adding adaptivity to our calculations makes our semi-implicit methods even more efficient. The need for the employment of the AMR technology in our computations is as follows. When solving underwater explosion and implosion problems, it is important to resolve the flow only around the high gradient regions such as shocks or material discontinuities in order to obtain efficient and accurate solution representations and save CPU time. Automating dynamic Adaptive Mesh Refinement techniques are customized to serve these purposes. The idea behind the Adaptive Mesh Refinement technique [5,4] is to overlay successively finer resolution grid patches on top of underlying coarse grids, and to introduce a recursive time integration algorithm, then to synchronize the data between different grid levels. The time integration algorithm can be applied in two ways. One way is that all grid levels are advanced with the finest grid level time step. This is called the *no-time-subcycling* procedure. Another way is that a coarser grid level is advanced with a coarse time step and a finer grid level is advanced with multiple fine time steps until the finer level time reaches to the coarser level time. This is called the *time-subcycling* procedure.

Early Adaptive Mesh Refinement (AMR) techniques [5,4,3] are developed for solving hyperbolic conservation laws. Later, they are extended to solve the compressible Navier–Stokes equations [18,17]. There have been significant efforts to solve incompressible or weakly compressible flows adaptively [1,15,11,13,2,8,19].

Our method shares some common features with [1] in the sense that we both subcycle in time and we both provide velocity continuity across the coarse/fine grids. Almgren et al. [1] method includes a synchronization projection step to provide velocity and pressure continuity across the coarse/fine grids. In [1], firstly, the velocity field is advanced on each level separately allowing velocity mismatch across the coarse/fine levels, then a multi-level composite projection step is applied at the end of each coarse level to correct the velocity differences. As a result their correction procedure modifies the solution on both coarse and fine levels. On the other hand, during our time advancing step, we solve the pressure equation on the current level and all levels above simultaneously. This produces accurate pressure boundary conditions for the next finer level. Also, since we solve on all levels $l' \geqslant l$ simultaneously, the velocity mismatch error, that gets corrected at the end of the ensuing fine time level integration steps, is significantly reduced. During the synchronization step, we solve the synchronization equations in the underlying coarse regions with Neumann boundary conditions. In this way, we maintain the velocity continuity without modifying the solution at the fine levels, i.e., we assume that the fine level velocity is correct and should not be changed.

The contents of the present paper is as follows. In Section 2, we describe the governing dynamics equations. In Section 3, we review the AMR grid hierarchy and define the components of our adaptive semi-implicit algorithms. In Section 4, we present the numerical results from the computations of underwater explosion/implosion test cases. Section 5 includes some concluding remarks.