Journal of Computational Physics 296 (2015) 241–262

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A time semi-implicit scheme for the energy-balanced coupling of a shocked fluid flow with a deformable structure

Maria Adela Puscas [a,b,c,*], Laurent Monasse [a], Alexandre Ern [a], Christian Tenaud [c], Christian Mariotti [b], Virginie Daru [c,d]

[a] Université Paris-Est, CERMICS (ENPC), 77455 Marne la Vallée cedex, France
[b] CEA-DAM-DIF, 91297 Arpajon, France
[c] LIMSI-CNRS, Bât 508 Campus Universitaire, Rue John Von Neumann, 91403 Orsay Cedex, France
[d] Lab. DynFluid, Ensam, 75013 Paris, France

ARTICLE INFO

Article history:
Received 20 May 2014
Received in revised form 9 March 2015
Accepted 11 April 2015
Available online 29 April 2015

Keywords:
Fluid–structure interaction
Finite volume
Immersed boundary
Conservative method
Energy preservation

ABSTRACT

The objective of this work is to present a conservative coupling method between an inviscid compressible fluid and a deformable structure undergoing large displacements. The coupling method combines a cut-cell Finite Volume method, which is exactly conservative in the fluid, and a symplectic Discrete Element method for the deformable structure. A time semi-implicit approach is used for the computation of momentum and energy transfer between fluid and solid, the transfer being exactly balanced. The coupling method is exactly mass-conservative (up to round-off errors in the geometry of cut-cells) and exhibits numerically a long-time energy-preservation for the coupled system. The coupling method also exhibits consistency properties, such as conservation of uniform movement of both fluid and solid, absence of numerical roughness on a straight boundary, and preservation of a constant fluid state around a wall having tangential deformation velocity. The performance of the method is assessed on test cases involving shocked fluid flows interacting with two and three-dimensional deformable solids undergoing large displacements.

1. Introduction

Fluid–structure interaction phenomena occur in many fields, such as aeronautics, civil engineering, energetics, biology, and in the military and safety domains. In this context for instance, the effects of an explosion on a building involve complex non-linear phenomena (shock waves, cracking, fragmentation, etc.) [1,2], and the characteristic time scales of these phenomena are extremely short. The driving effect of the fluid–structure interaction is the fluid overpressure, and viscous effects play a lesser role. With an eye toward these applications, we consider an inviscid compressible flow with shock waves interacting with a deformable solid object.

Numerical methods for fluid–structure interaction can be broadly categorized into monolithic and partitioned methods. In monolithic (Eulerian [3,4] or Lagrangian [5,6]) methods, the fluid and the solid equations are solved simultaneously

* Corresponding author at: Université Paris-Est, CERMICS (ENPC), 77455 Marne la Vallée Cedex, France.
E-mail addresses: puscasa@cermics.enpc.fr, adela.puscas@cea.fr, adela.puscas@limsi.fr, adelapuscas@gmail.com (M.A. Puscas), monassel@cermics.enpc.fr (L. Monasse), ern@cermics.enpc.fr (A. Ern), christian.tenaud@limsi.fr (C. Tenaud), christian.mariotti@cea.fr (C. Mariotti), virginie.daru@limsi.fr, virginie.daru@ensam.eu (V. Daru).

http://dx.doi.org/10.1016/j.jcp.2015.04.012
0021-9991/

at each time step. However, in many numerical schemes, the fluid is described in Eulerian formulation and the solid in Lagrangian formulation. This is possible in partitioned approaches where the fluid and the solid equations are solved separately, and an interface module is used to exchange information between the fluid and the solid solvers to enforce the dynamic boundary conditions at their common interface. Two main types of methods have been developed in this context: Arbitrary Lagrangian–Eulerian (ALE) methods [7,8] and fictitious domain methods [9–19]. The ALE method hinges on a mesh fitting the solid boundary, and therefore requires remeshing of the fluid domain when the solid goes through large displacements and topological changes due to fragmentation. Instead, fictitious domain methods, as those considered herein, work on a fixed fluid grid to which the solid is superimposed, and additional terms are introduced in the fluid formulation to impose the boundary conditions at the fluid–solid interface.

Conservative cut-cell Finite Volume methods for compressible fluid–structure interaction have been proposed by Noh [19]. Therein, a Lagrangian method for the solid is coupled with an Eulerian Finite Volume method for the compressible flow satisfying mass, momentum, and energy conservation in the fluid. Such methods have been used in a number of applications [10,11,14,15,19,20]. A coupling method between an inviscid compressible fluid and a rigid body undergoing large displacements has been developed in [21,22] using a cut-cell Finite Volume method. The coupling method is conservative in the sense that (i) mass, momentum, and energy conservation in the fluid is achieved by the cut-cell Finite Volume method as in [19], and (ii) the momentum and energy exchange between the fluid and the solid is balanced. As a result, the system is exactly conservative, up to round-off errors in the geometry of cut-cells. Moreover, the coupling method exhibits interesting consistency properties, such as conservation of uniform movement of both fluid and solid, and absence of numerical roughness on a straight boundary.

The main purpose of this work is to develop a three-dimensional conservative coupling method between a compressible inviscid fluid and a deformable solid undergoing large displacements. By conservative, we mean that properties (i) and (ii) above are satisfied, as in [21,22], and additionally that a symplectic scheme is used for the Lagrangian solid ensuring the conservation of a discrete energy (which is a close approximation of the physical energy). As a result, the coupled discrete system is not exactly energy-conservative, but we show numerically that our strategy yields long-time energy-preservation for the coupled system. Furthermore, as in [21,22], the Finite Volume method for the fluid is high-order in smooth flow regions and away from the solid boundary, while it is first-order near the shocks (due to the flux limiters) and in the vicinity of the solid boundary. Consequently, the coupling method is overall first-order accurate. Still, the use of a high-order method in smooth regions is useful to limit numerical diffusion in the fluid, as discussed in [23]. In any case, the coupling method, which is the focus of this work, is independent of the choice of the fluid fluxes.

While the core of the present method hinges on the techniques of [22] for a rigid solid, many new aspects have to be addressed. A reconstruction of the solid boundary around the solid assembly is needed since the solid deforms through the interaction with the fluid. Furthermore, a time semi-implicit scheme is introduced for the momentum and energy exchange, so as to take into account the deformation of the solid boundary during the time step. The advantage of this scheme with respect to an explicit one is to achieve additional consistency properties, such as the absence of pressure oscillations near a solid wall having only tangential deformation. The time semi-implicit scheme evaluates the fluid fluxes as well as the solid forces and torques only once per time step, which is important for computational efficiency of the scheme. Additionally, we prove that the time semi-implicit scheme converges with geometric rate under a CFL condition, which, under the assumption that the solid density is larger than the fluid density, is less restrictive than the fluid CFL condition.

The paper is organized as follows: Section 2 briefly describes the basic ingredients (which are common with [22]): the fluid and solid discretization methods and the cut-cell Finite Volume method. Section 3 presents the conservative coupling method based on the time semi-implicit procedure. Section 4 discusses several properties of the coupling method. Section 5 presents numerical results on strong fluid discontinuities interacting with two and three-dimensional deformable solids undergoing large displacements. Section 6 contains concluding remarks. Finally, Appendix A provides some background on the Discrete Element method used to discretize the solid, and Appendix B contains the convergence proof for the time semi-implicit scheme.

2. Basic ingredients

2.1. Fluid discretization

For inviscid compressible flow, the fluid state is governed by the Euler equations, which can be written in conservative form as

$$\frac{\partial}{\partial t}U + \frac{\partial}{\partial x}F(U) + \frac{\partial}{\partial y}G(U) + \frac{\partial}{\partial z}H(U) = 0, \tag{1}$$

where $U = (\rho, \rho u, \rho v, \rho w, \rho E)^t$ is the conservative variable vector and $F(U)$, $G(U)$, and $H(U)$ indicate the inviscid fluxes

$$F(U) = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \\ \rho u w \\ (\rho E + p)u \end{pmatrix}, \; G(U) = \begin{pmatrix} \rho v \\ \rho u v \\ \rho v^2 + p \\ \rho v w \\ (\rho E + p)v \end{pmatrix}, \; H(U) = \begin{pmatrix} \rho w \\ \rho u w \\ \rho v w \\ \rho w^2 + p \\ (\rho E + p)w \end{pmatrix},$$