Journal of Computational Physics 307 (2016) 670–695

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A cut-cell finite volume – finite element coupling approach for fluid–structure interaction in compressible flow

Vito Pasquariello [a,*], Georg Hammerl [b], Felix Örley [a], Stefan Hickel [a,c], Caroline Danowski [b], Alexander Popp [b], Wolfgang A. Wall [b], Nikolaus A. Adams [a]

[a] *Institute of Aerodynamics and Fluid Mechanics, Technische Universität München, Boltzmannstr. 15, 85748 Garching bei München, Germany*
[b] *Institute for Computational Mechanics, Technische Universität München, Boltzmannstr. 15, 85748 Garching bei München, Germany*
[c] *Aerodynamics Group, Faculty of Aerospace Engineering, Technische Universiteit Delft, Kluyverweg 1, 2629 HS Delft, The Netherlands*

ARTICLE INFO

Article history:
Received 11 January 2015
Received in revised form 18 September 2015
Accepted 7 December 2015
Available online 15 December 2015

Keywords:
Fluid–structure interaction
Compressible flow
Cut-cell method
Immersed boundary method
Mortar method

ABSTRACT

We present a loosely coupled approach for the solution of fluid–structure interaction problems between a compressible flow and a deformable structure. The method is based on staggered Dirichlet–Neumann partitioning. The interface motion in the Eulerian frame is accounted for by a conservative cut-cell Immersed Boundary method. The present approach enables sub-cell resolution by considering individual cut-elements within a single fluid cell, which guarantees an accurate representation of the time-varying solid interface. The cut-cell procedure inevitably leads to non-matching interfaces, demanding for a special treatment. A Mortar method is chosen in order to obtain a conservative and consistent load transfer. We validate our method by investigating two-dimensional test cases comprising a shock-loaded rigid cylinder and a deformable panel. Moreover, the aeroelastic instability of a thin plate structure is studied with a focus on the prediction of flutter onset. Finally, we propose a three-dimensional fluid–structure interaction test case of a flexible inflated thin shell interacting with a shock wave involving large and complex structural deformations.

1. Introduction

Compressible fluid–structure interaction (FSI) occurs in a broad range of technical applications involving, e.g., nonlinear aeroelasticity [16,42] and shock-induced deformations of rocket nozzles [23,55]. The numerical modeling and simulation of compressible FSI can be challenging, in particular if an accurate representation of the structural interface within the fluid solver and a consistent coupling of both subdomains is required.

FSI algorithms are generally classified as monolithic or partitioned. One main advantage often attributed to monolithic approaches is their numerical robustness due to solving a single system which includes the full information of the coupled nonlinear FSI problem. On the other hand, partitioned algorithms for FSI are often used because they facilitate the coupling of different specialized single-field solvers. A further distinction can be made between loosely and strongly coupled algorithms, depending on whether the coupling conditions are satisfied exactly at each time step, or not. While partitioned algorithms can be made strong by introducing equilibrium iterations [34], loosely coupled approaches are more frequently used in the field of aeroelasticity and compressible flows in general [6,16]. A disadvantage of loosely coupled partitioned algorithms is the artificial added mass effect [5,21], which may lead to numerical instability in incompressible flows and for

* Corresponding author. Tel.: +49 89 289 16134; fax: +49 89 289 16139.
E-mail address: vito.pasquariello@tum.de (V. Pasquariello).

http://dx.doi.org/10.1016/j.jcp.2015.12.013
0021-9991/

high fluid–solid density ratios. Recently, so-called Added-Mass Partitioned algorithms have been developed for compressible fluids interacting with rigid and elastic solids [1,3] as well as for incompressible fluids [2]. These methods allow to overcome the added mass instability by formulating appropriate fluid–structure interface conditions.

FSI problems involve a load and motion transfer at the conjoined interface. In the simple case of matching fluid and solid discretization, this results in a trivial task. However, different resolution requirements within the fluid and solid fields lead to non-matching discrete interfaces. An overview of existing coupling methods for non-matching meshes can be found in [9]. Simple methods such as nearest-neighbor interpolation and projection methods are frequently used [17,31]. The mentioned methods do not conserve angular momentum across the interface. Consistency can be achieved with more sophisticated approaches, such as weighted residual methods, which introduce Lagrange multipliers as additional interface variables. In this context, Mortar methods have first been proposed for non-overlapping domain decomposition in [4], enhanced with dual shape functions for the Lagrange multipliers in [53] and applied to FSI problems and mesh tying in fluid flow, e.g. in [13,33]. While Mortar methods introduce Lagrange multipliers only on one side of the interface, Localized Lagrange Multipliers consider them on both sides of the interface [47].

Another classification of FSI methods is based on the representation of the time-varying solid interface within the fluid domain. Two main approaches can be distinguished in this context, which are Arbitrary Lagrangian Eulerian (ALE) methods [10,18], and Immersed Boundary Methods (IBM) [38,41]. ALE approaches employ body-fitted grids, hence requiring a mesh evolution algorithm. This task may be complex in case of large solid displacements. On the other hand, IBM often operate on fixed Cartesian fluid grids, making this type of approach very appealing for the simulation of flows past complex geometries and for the solution of FSI problems with large deformations. IBM, such as continuous forcing and ghost-cell approaches, may suffer from spurious loss or production of mass, momentum and energy at the interface [38]. Such non-conservativity poses a particular problem for large-eddy simulations, which employ coarse grids and rely on an accurate flow prediction in near-wall regions over large time scales. Moreover, the accurate capturing of shocks is based on conservation properties. Conservativity is recovered with Cartesian cut-cell methods, which were first introduced by Clarke et al. [7] and Gaffney and Hassan [22] for inviscid flows and later extended to viscous flows by Udaykumar et al. [52] and Ye et al. [54]. In this method, the finite volume cells at the boundaries are reshaped to fit locally the boundary surface with a sharp interface, which in turn assures strict conservation of mass, momentum and energy. A drawback of cut-cell methods is that the fluid volume fraction of cut-cells may become very small and therefore can lead to numerical instability with explicit time integration schemes. A stabilization of the underlying time integration scheme can be achieved by so-called cell-merging [54], cell-linking [32] or flux redistribution techniques [8,30].

In this paper we develop a loosely coupled approach for the solution of FSI problems between a compressible fluid and a deformable structure. We employ the Finite Volume Method (FVM) for solving the Euler equations on Cartesian grids and the Finite Element Method (FEM) for solving the structural problem. The interface motion is accounted for by a conservative cut-cell IBM. Previous proposed methods reconstruct the interface geometry based on a level-set function [26,27,36]. Örley et al. [40] developed a conservative cut-element method that allows for representing the fluid–solid interface with sub-cell resolution for rigid body motion. We extend this method to arbitrary interface deformations. The combination of a cut-element IBM with a Mortar method for coupling of the solid and fluid subdomains in a consistent and efficient way is the essential new contribution of this paper.

This paper is structured as follows: First, the governing equations for fluid and solid and the fluid–structure interface conditions are introduced in Section 2. Section 3 gives a detailed overview on the numerical treatment of moving boundaries together with the discretization methods used for the fluid. The FEM used to solve the structural problem is presented in Section 4. In Section 5, the staggered coupling algorithm is presented together with the new coupling approach for non-matching interfaces. In Section 6, the method is validated with well-established two-dimensional test cases and a convergence study is presented. In Section 7, we propose a new test case for the interaction between a flexible inflated thin shell and a shock wave, demonstrating in particular the capability of our FSI approach to handle large three-dimensional deformations. Concluding remarks are given in Section 8.

2. Mathematical and physical model

As depicted in Fig. 1, the computational domain is divided into a fluid and solid domain, Ω_F and Ω_S, respectively. The conjoined interface is denoted as $\Gamma = \Omega_\mathsf{F} \cap \Omega_\mathsf{S}$ and its normal vector $\boldsymbol{n}^\Gamma$ in spatial configuration points from the solid into the fluid domain.

2.1. Governing equations for the fluid

We consider the three-dimensional, fully compressible Euler equations in conservative form

$$\frac{\partial \boldsymbol{w}}{\partial t} + \nabla \cdot \boldsymbol{K}(\boldsymbol{w}) = 0 \text{ in } \Omega_\mathsf{F}\,. \tag{1}$$

The state vector $\boldsymbol{w} = [\rho_\mathsf{F}, \rho_\mathsf{F} u_1, \rho_\mathsf{F} u_2, \rho_\mathsf{F} u_3, E_\mathrm{t}]$ contains the conserved variables density ρ_F, momentum $\rho_\mathsf{F}\boldsymbol{u}$ and total energy E_t. The subscript F denotes fluid quantities and is used whenever a distinction between both subdomains is necessary. The individual contributions of the flux tensor $\boldsymbol{K} = (\boldsymbol{f}, \boldsymbol{g}, \boldsymbol{h})$ are given as