ARTICLE IN PRESS

Journal of Computational Physics xxx (2012) xxx–xxx

Contents lists available at SciVerse ScienceDirect

Journal of Computational Physics

journal homepage: www.elsevier.com/locate/jcp

An accurate moving boundary formulation in cut-cell methods

Lennart Schneiders [a,*], Daniel Hartmann [b], Matthias Meinke [a], Wolfgang Schröder [a]

[a] Institute of Aerodynamics, RWTH Aachen University, Wüllnerstr. 5a, Aachen 52062, Germany
[b] Division of Engineering and Applied Science, California Institute of Technology, Pasadena, CA 91125, USA

ARTICLE INFO

Article history:
Received 10 September 2011
Received in revised form 6 August 2012
Accepted 20 September 2012
Available online xxxx

Keywords:
Cut-cell method
Immersed boundary method
Moving boundary
Spurious force oscillations
Oscillating cylinder
Compressible Navier–Stokes

ABSTRACT

A cut-cell method for Cartesian meshes to simulate viscous compressible flows with moving boundaries is presented. We focus on eliminating unphysical oscillations occurring in Cartesian grid methods extended to moving-boundary problems. In these methods, cells either lie completely in the fluid or solid region or are intersected by the boundary. For the latter cells, the time dependent volume fraction lying in the fluid region can be so small that explicit time-integration schemes become unstable and a special treatment of these cells is necessary. When the boundary moves, a fluid cell may become a cut cell or a solid cell may become a small cell at the next time level. This causes an abrupt change in the discretization operator and a suddenly modified truncation error of the numerical scheme. This temporally discontinuous alteration is shown to act like an unphysical source term, which deteriorates the numerical solution, i.e., it generates unphysical oscillations in the hydrodynamic forces exerted on the moving boundary. We develop an accurate moving boundary formulation based on the varying discretization operators yielding a cut-cell method which avoids these discontinuities. Results for canonical two- and three-dimensional test cases evidence the accuracy and robustness of the newly developed scheme.

1. Introduction

Flows with moving boundaries are of significant importance in many technical, biological, and geophysical applications. These applications range, for instance, from the prediction of wave loading on offshore structures to the study of bubble dynamics, from the control of aeroelastic response to understanding aquatic locomotion, and from the simulation of glacial motion to the design of artificial heart valves. In recent years, the numerical simulation of related problems of fluid–structure interaction as well as forced structural motion became more and more of interest. While the growing computational resources allow to tackle increasingly complex flow configurations involving large structural displacements and multiple independent boundaries, numerical methods which enable an easy handling of this complexity become especially attractive. Cartesian grid methods generally allow a fast and automated grid generation, which otherwise can become time-consuming and tedious.

It goes without saying that boundary-conforming grid methods have successfully been applied to moving-boundary problems by several authors (e.g. [1,2]). However, especially when large structural motion and deformation are considered or multiple immersed bodies move independently from each other, the required deformation of these meshes usually deteriorates their quality. In this case, a re-meshing along with a projection of the solution onto the new grid is necessary. This procedure can dominate the overall computational cost of the solver and may decrease its accuracy.

* Corresponding author. Tel.: +49 241 80 90419; fax: +49 241 80 92257.
E-mail addresses: office@aia.rwth-aachen.de, L.Schneiders@aia.rwth-aachen.de (L. Schneiders).

0021-9991/$ - see front matter © 2012 Elsevier Inc. All rights reserved.
http://dx.doi.org/10.1016/j.jcp.2012.09.038

Cartesian grid methods use non-boundary-conforming meshes that are not directly affected by the complexity of the embedded boundaries. Their regular structure allows for a simple and automated grid generation, fast solution algorithms, and a conceptionally simple decomposition of the grid for adaptation and parallel computing. This comes at the cost of efficiency for resolving boundary layer flows at very high Reynolds numbers [3]. Concerning the tracking and handling of the moving boundary several approaches exist. The classical immersed-boundary method of Peskin [4] was motivated by the desire to simulate the flow in the cardiovascular system involving elastic muscular boundaries. The elastic boundary is modeled by adding an external force field to the continuous equations of fluid motion. Since the nodes of the Cartesian grid usually do not coincide with the location of the immersed boundary the forcing terms are smeared across a few grid cells in the vicinity of the boundary using smoothed delta functions. This generally renders the approach a first-order method at the boundary which is why it is also known as the diffused-interface method. Different immersed boundary formulations have been proved to be robust for moving boundary problems, e.g. [5,6]. In [7] an immersed-boundary method with direct forcing has been introduced which allows to treat the boundary as a sharp interface. In this method, the forcing terms are added to the discretized equations and determined from the discrete application of the boundary conditions. There is a variety of other sharp interface methods that have been used for moving-boundary problems. Among these are finite difference approaches [8,9], the ghost fluid method [10,11], and the immersed interface method [12,13]. However, none of the above approaches strictly satisfies the conservation laws near the boundary. Unlike these methods, global and local conservation can be achieved by the Cartesian cut-cell method for moving-boundary problems [14–17]. This method is of the finite-volume type, wherein the cut cells that are intersected by the moving boundary are reshaped to locally fit the boundary surface. The method exactly accounts for the forces at the moving boundary and strictly conserves mass, momentum, and energy. This makes it especially attractive for fluid–structure interaction problems [18]. However, particular effort is necessary for a robust and accurate discretization and to alleviate stability issues arising from cut cells, which may become arbitrarily small in the process of reshaping. While Cartesian methods for compressible flow have been applied to moving-boundary problems by several authors [10,11,19,20], to the best of the authors' knowledge, the use of the cut-cell method has been only reported for two-dimensional problems [14,16] or 3D inviscid flow [15,21].

In this paper, we develop a Cartesian cut-cell method for viscous compressible three-dimensional flow with moving boundaries. In particular, we show how to avoid numerical oscillations in the boundary force that are observed when the cut-cell method for fixed geometries proposed in [22] is directly extended to account for moving boundaries in an inertial reference frame. The appearance of these oscillations has recently been discussed by our group and traced back to the frequent changes of the discrete operators near the moving boundaries [23,24]. Similar oscillations were also reported on in simulations of incompressible flow using the direct forcing immersed-boundary method [25–32]. Although numerous articles were published discussing the direct forcing approach for moving boundary problems during the last decade the origin of these oscillations is still not fully understood. Uhlmann [26] and Yang et al. [28] show that increasing the support of the discrete delta functions can smoothen these oscillations. Yang et al. [28] argue that the oscillations are induced by delta functions, the derivatives of which do not satisfy certain moment conditions. Lee et al. [31] consider temporal and spatial discontinuities as a source for the oscillations and investigate the influence of different grid widths and time step sizes. Seo and Mittal [32] show that spurious pressure oscillations are caused by a violation of the geometric conservation law at the immersed boundary. To reduce the oscillations they ensure geometric conservation by making partial use of the cut-cell method and its conservation properties.

Moreover, we demonstrate that for compressible flow problems such oscillations can also occur in the Cartesian cut-cell method in combination with, e.g., the widely-used cell-merging technique despite it being conservative. We show that spurious oscillations in the fluid force exerted on an immersed boundary are caused by abrupt changes of the spatial discretization operator, which occur when the immersed boundary moves relative to the computing grid. Based on the Cartesian cut-cell formulation for fixed boundaries by Hartmann et al. [22], we develop a novel robust method for moving boundaries which significantly reduces the non-physical oscillations. The main feature of the method is a discretization which is formulated such that it responds smoothly to abrupt changes of the mesh topology involved in moving-boundary computations, while retaining the accuracy and conservation properties of the traditional cut-cell method. The moving boundaries are represented using a level-set framework which generally allows a flexible and sharp description of the boundary motion.

The remainder of this paper is organized as follows. In Section 2, the governing equations are summarized. The numerical method is described in Section 3, while in Section 4 the novel discretization scheme for moving boundaries is presented. In Section 5, the method is validated for two- and three-dimensional test cases, evidencing its convincing results with respect to established data from the literature. Finally, a brief summary is given in Section 6.

2. Mathematical model

2.1. Governing equations and notation

Consider a rigid or deformable solid $\{\mathbf{x}|\mathbf{x} \in \Omega^s(t)\}$ bounded by the surface $\Gamma(t) = \partial\Omega^s(t)$, which moves through a viscous compressible fluid in the domain Ω at time t. The fluid covers the space $\Omega^f(t) = \Omega \setminus \Omega^s(t)$. The local boundary velocity is denoted by $\mathbf{u}_\Gamma(\mathbf{x}, t), \mathbf{x} \in \Gamma(t)$. The number of space dimensions will be denoted by the quantity d.

Please cite this article in press as: L. Schneiders et al., An accurate moving boundary formulation in cut-cell methods, J. Comput. Phys. (2012), http://dx.doi.org/10.1016/j.jcp.2012.09.038