Journal of Computational Physics 231 (2012) 2892–2923

Contents lists available at SciVerse ScienceDirect

Journal of Computational Physics

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

A systematic approach for constructing higher-order immersed boundary and ghost fluid methods for fluid–structure interaction problems

Xianyi Zeng [a], Charbel Farhat [a,b,c,*]

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

ARTICLE INFO

Article history:
Received 28 September 2011
Received in revised form 16 December 2011
Accepted 26 December 2011
Available online 4 January 2012

Keywords:
Embedded boundary method
Fluid structure interaction
Ghost fluid method
Higher-order
Immersed boundary method
Operator matching
Surrogate interface

ABSTRACT

A systematic approach is presented for constructing higher-order immersed boundary and ghost fluid methods for CFD in general, and fluid–structure interaction problems in particular. Such methods are gaining popularity because they simplify a number of computational issues. These range from gridding the fluid domain, to designing and implementing Eulerian-based algorithms for challenging fluid–structure applications characterized by large structural motions and deformations or topological changes. However, because they typically operate on non body-fitted grids, immersed boundary and ghost fluid methods also complicate other issues such as the treatment of wall boundary conditions in general, and fluid–structure transmission conditions in particular. These methods also tend to be at best first-order space-accurate at the immersed interfaces. In some cases, they are also provably inconsistent at these locations. A methodology is presented in this paper for addressing this issue. It is developed for inviscid flows and prescribed structural motions. For the sake of clarity, but without any loss of generality, this methodology is described in one and two dimensions. However, its extensions to flow-induced structural motions and three dimensions are straightforward. The proposed methodology leads to a departure from the current practice of populating ghost fluid values independently from the chosen spatial discretization scheme. Instead, it accounts for the pattern and properties of a preferred higher-order discretization scheme, and attributes ghost values as to preserve the formal order of spatial accuracy of this scheme. It is illustrated in this paper by its application to various finite difference and finite volume methods. Its impact is also demonstrated by one- and two-dimensional numerical experiments that confirm its theoretically proven ability to preserve higher-order spatial accuracy, including in the vicinity of the immersed interfaces.

1. Introduction

Eulerian immersed boundary and ghost fluid methods (for example, see [1–4] and references cited therein) are usually preferred over alternative approaches based, for example, on the Arbitrary Lagrangian Eulerian framework [5,6], for the solution of fluid–structure interaction (FSI) problems characterized by large structural motions and/or deformations or by topological changes. Such methods allow a rigid or flexible moving body to penetrate the CFD (computational fluid dynamics) grid. For this reason, they operate on non body-fitted grids. They typically distinguish between "real" (or "active") fluid grid

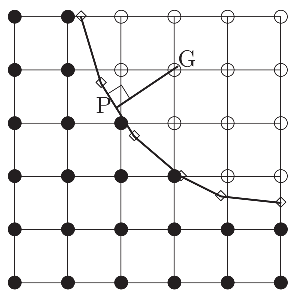
* Corresponding author at: Department of Mechanical Engineering, Stanford University, Stanford, CA 94305-4035, USA. Tel.: +1 650 723 3840; fax: +1 650 725 3525.
E-mail address: cfarhat@stanford.edu (C. Farhat).

0021-9991/$ - see front matter © 2012 Elsevier Inc. All rights reserved.
doi:10.1016/j.jcp.2011.12.027

points lying in the physical fluid domain, and ‘‘ghost’’ (or ‘‘inactive’’ or ‘‘fictitious’’) fluid grid points lying inside the moving obstacle. This distinction is usually a dynamic one because FSI problems are typically dynamic problems. When at the end of a computational time-instance t^n the status of a fluid grid point changes from real to ghost — which is referred to in this paper as the RTG scenario — the value of the fluid state vector at this ghost point must be provided in order to enable the advancement of the flow computation to the next time-instance t^{n+1}. Similarly, when at the end of t^n the status of a fluid grid point changes from ghost to real — which is referred to here as the GTR scenario — a new value of the fluid state vector must be provided at this grid point.

For the RTG scenario, most approaches proposed in the literature fall into two categories. The first one consists of methods which project a ghost fluid grid point onto the fluid–structure interface (see Fig. 1.1), interpolate the velocity of the structure at the projection point, and set the value of the fluid velocity vector at the ghost fluid grid point (or the normal component of this vector when the flow is assumed to be inviscid) to this interpolated value (or its normal component for inviscid flows) [7,2]. This enforces an approximate form of the non-penetration condition at the fluid–structure interface. In this case, the remaining primal variables of the fluid state vector at the ghost fluid grid point (density, pressure, or entropy) are computed by first approximating their counterparts at the projection point using data from nearby real fluid grid points and either interpolation or extrapolation, then extrapolating the obtained values to the location of that ghost fluid grid point. Hence, this category of methods is easy to implement. However, it is only first-order accurate in space. For this reason, it is often equipped with adaptive mesh refinement (AMR) [8,2] in the vicinity of the fluid–structure interface. This improves its practical accuracy, but at the cost of an increase of its implementational (and computational) complexity, especially in three dimensions (3D). The methods in the second category [4,9] can be labeled as ‘‘mirroring’’ methods. Not only they project a ghost fluid grid point of interest onto the fluid–structure interface, but they also determine its reflection with respect to the tangential interface — a line in two dimensions (2D) and a plane in three dimensions (3D) — containing the projection point. Since a mirroring point usually falls in the physical fluid domain, the fluid state variables at this point are computed by interpolation. Then, the fluid state vector at the ghost fluid grid point is computed as follows. The fluid velocity vector (or its normal component in the case of an inviscid flow) is obtained by linearly extrapolating the interpolated fluid velocity vector at the mirroring point and the structural velocity vector at the projection point (see Fig. 1.2). The values of the remaining primal fluid variables at the ghost fluid grid point are set to the interpolated values at the mirroring real fluid grid point. In theory, the spatial accuracy of this second category of methods is one order higher than that of the first one described above. However, to the best of knowledge of the authors, second-order spatial accuracy has been reported for such methods only for the case of a stationary fluid–structure interface [9]. For genuine FSI problems with dynamic fluid–structure interfaces, the mirroring procedure is sometimes combined with local grid refinement (for example, see [4]) to enhance the delivered spatial accuracy in the vicinity of the fluid–structure interface.

For the GTR scenario, two different computational strategies can be found in the literature. In the first one, the fluid state vector at a fluid grid point which switches from the ghost to the real status at the end of time-instance t^n is determined from



• Real fluid grid point   G: Ghost fluid grid point

○ Ghost fluid grid point   P: Projection of G onto interface

◇ Structure node   M: Mirroring grid point of G

$\vec{V}_n$: Normal velocity vector p : Fluid pressure

$\vec{V}_t$: Tangential velocity vector ρ : Fluid density

$_{,A}$: Variable at point A $^{S/F}$: Structure/fluid variable

$\vec{V}_{n,G} = \vec{V}^S_{n,P}$ Interpolated from values at structure nodes

$\vec{V}_{t,G} = \vec{V}^F_{t,P}$ Interpolated from values at real fluid grid points

$\rho_G = \rho^F_P$ Interpolated from values at real fluid grid points

$p_G = p^F_P$ Interpolated from values at real fluid grid points

Fig. 1.1. Projection approach (2D illustration).