Journal of Computational Physics 365 (2018) 74–104

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A family of position- and orientation-independent embedded boundary methods for viscous flow and fluid–structure interaction problems

Daniel Z. Huang [a], Dante De Santis [d], Charbel Farhat [b,c,a,*]

[a] Institute for Computational and Mathematical Engineering
[b] Department of Aeronautics and Astronautics
[c] Department of Mechanical Engineering, Stanford University, Stanford, CA 94305-4035, USA
[d] Nuclear Research and Consultancy Group, Westerduinweg 3, 1755 LE Petten, The Netherlands

ARTICLE INFO

Article history:
Received 18 December 2017
Received in revised form 17 March 2018
Accepted 17 March 2018
Available online 22 March 2018

Keywords:
Compressible flows
Embedded boundary method
Finite volume method
Fluid–structure interaction
Immersed boundary method
Turbulent flows

ABSTRACT

The Finite Volume method with Exact two-material Riemann Problems (FIVER) is both a computational framework for multi-material flows characterized by large density jumps, and an Embedded Boundary Method (EBM) for computational fluid dynamics and highly nonlinear Fluid–Structure Interaction (FSI) problems. This paper deals with the EBM aspect of FIVER. For FSI problems, this EBM has already demonstrated the ability to address viscous effects along wall boundaries, and large deformations and topological changes of such boundaries. However, like for most EBMs – also known as immersed boundary methods – the performance of FIVER in the vicinity of a wall boundary can be sensitive with respect to the position and orientation of this boundary relative to the embedding mesh. This is mainly due to ill-conditioning issues that arise when an embedded interface becomes too close to a node of the embedding mesh, which may lead to spurious oscillations in the computed solution gradients at the wall boundary. This paper resolves these issues by introducing an alternative definition of the active/inactive status of a mesh node that leads to the removal of all sources of potential ill-conditioning from all spatial approximations performed by FIVER in the vicinity of a fluid–structure interface. It also makes two additional contributions. The first one is a new procedure for constructing the fluid–structure half Riemann problem underlying the semi-discretization by FIVER of the convective fluxes. This procedure eliminates one extrapolation from the conventional treatment of the wall boundary conditions and replaces it by an interpolation, which improves robustness. The second contribution is a post-processing algorithm for computing quantities of interest at the wall that achieves smoothness in the computed solution and its gradients. Lessons learned from these enhancements and contributions that are triggered by the new definition of the status of a mesh node are then generalized and exploited to eliminate from the original version of the FIVER method its sensitivities with respect to both of the position and orientation of the wall boundary relative to the embedding mesh, while maintaining the original definition of the status of a mesh node. This leads to a family of second-generation FIVER methods whose performance is illustrated in this paper for several flow and FSI problems. These include a challenging flow problem over

* Corresponding author at: Department of Aeronautics and Astronautics, Stanford University, Stanford, CA 94305-4035, USA.
E-mail address: cfarhat@stanford.edu (C. Farhat).

https://doi.org/10.1016/j.jcp.2018.03.028

a bird wing characterized by a feather-induced surface roughness, and a complex flexible flapping wing problem for which experimental data is available.

1. Introduction

Embedded or Immersed Boundary Methods (EBMs or IBMs) continue to gain popularity for the solution of Computational Fluid Dynamics (CFD) and Fluid–Structure Interaction (FSI) problems [1–13]. Such methods can be described as Eulerian CFD and FSI methods. They compute flows on non-body-fitted CFD meshes in which discrete representations of wet surfaces of obstacles are embedded or immersed. They are also known under other names, including fictitious domain [14] and Cartesian [2] methods. For CFD problems in general, they are attractive because they introduce a high degree of automation in the task of mesh generation, and a significant flexibility in the meshing of complex geometries. It should be mentioned however that for turbulent flow computations at high Reynolds numbers, the benefits of this increased level of automation can be offset by the tendency of highly automated meshing processes to generate uniform elements around embedded discrete surfaces. Indeed, for the same mesh resolution near the wall boundaries, such a tendency leads to embedding meshes that are larger than counterparts that can be generated using well-designed body-fitted meshing methods. Nevertheless, in the presence of rough aerodynamic or hydrodynamic surface textures – such as those encountered, for example, in the geometrical descriptions of heat shields of re-entry vehicles, iced aircraft wings, feather-covered bird wings, and submarine hulls with biofouling – where the characteristic length scale of the roughness is much smaller than that of the wet surface of the obstacle, the highly automated nature of meshing for EBMs becomes a significant advantage over body-fitted meshing. By far however, the major advantages of EBMs are materialized for FSI applications characterized by large structural displacements, rotations, deformations, and/or topological changes [15,9,16]. For such applications, alternative methods based on, for example, Lagrangian [17,18] or Arbitrary Lagrangian–Eulerian (ALE) approaches [19–24] are either cumbersome or simply unfeasible. For example, Lagrangian approaches can be impractical for high-speed viscous flows. ALE approaches rely on mesh motion and deformation schemes [25–28]. Typically, these are unreliable for viscous FSI problems where the mesh elements in the vicinity of a fluid–structure interface – also referred to in the remainder of this paper as the material interface – are usually small and stretched. Furthermore, they lack robustness with respect to large structural motions and/or deformations, and are simply unfeasible in the event of topological changes such as those encountered during topology optimization [29] or those induced by crack propagation [13].

By definition, EBMs work with non-body-fitted meshes. Consequently, they incorporate a special treatment of the wall boundary (or transmission) conditions. For this purpose, some EBMs modify the stencil of the spatial approximation in the neighborhood of the embedded discrete surface [30]. Others rely for this purpose on forcing functions derived from physical arguments [1] or the adopted semi-discretization process [3]. Other similar methods perform the treatment of the wall boundary conditions based on the cut-cell methodology [31], where each mesh cell cut by the embedded discrete surface is reshaped to account only for its portion that lies within the physical fluid domain.

Recently, an alternative approach for designing EBMs that differs radically from those briefly characterized above was proposed in [9]. This approach is based on an idea that was first presented in [32] for the treatment of fluid–fluid interfaces in multi-phase flow problems, in the context of vertex-based Finite Volume (FV) methods. While developed primarily for compressible fluid flows, it is equally applicable to incompressible flows. It differentiates itself from other EBMs in three significant ways. First, it can operate on both structured and unstructured meshes. Second, it treats the velocity and pressure boundary conditions at the embedded discrete surface simultaneously, rather than disjointly. Third, instead of relying primarily on semi-empirical interpolation and/or extrapolation procedures for performing the treatment of the wall boundary conditions, it enforces the desired value of the fluid velocity at a point on a wall boundary and recovers the value of the fluid pressure at this location by solving there an appropriate, local, one-dimensional, exact, fluid–structure half Riemann problem. For this reason, this alternative approach for designing EBMs was named the Finite Volume method with Exact two-phase or two-material Riemann Problems, or simply FIVER.

FIVER was originally designed in the context of explicit time-discretizations for the solution of compressible, inviscid, multi-phase flow problems characterized by simple Equations of State (EOSs) but featuring large density jumps [32]. Then, it was generalized in [33] to arbitrary EOSs by incorporating in its core a computationally efficient sparse grid tabulation method for the numerical solution of one-dimensional, two-phase Riemann problems. Next, FIVER was extended in [9] to multi-material FSI problems by generalizing the concept of a one-dimensional two-phase Riemann problem to that of a local fluid–structure Riemann problem. This extension transformed FIVER into an Eulerian EBM for FSI that is capable of handling large structural deformations and arbitrary topological changes. Unlike many EBMs that operate only on Cartesian meshes [2,34], FIVER can operate on arbitrary meshes. This makes it particularly suitable for complex geometries and viscous flows. In [11], it was validated for the dynamic implosion of underwater cylindrical shells. Specifically, FIVER was reported in [11] to reproduce with good accuracy the large deformations of collapsing cylindrical structures and the compression waves that emanate from them. In [35], this method was extended to implicit time-discretizations. In [36,37], it was generalized to turbulent viscous FSI problems, and in [13], it was extended again to the solution of FSI problems with dynamic fracture. In [38], FIVER was first shown to be – like many other EBMs – inconsistent in the neighborhood of a material interface, and