Computers & Fluids 90 (2014) 21–41

Contents lists available at ScienceDirect

Computers & Fluids

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

A second order penalized direct forcing for hybrid Cartesian/immersed boundary flow simulations

C. Introïni [a], M. Belliard [a,*], C. Fournier [b]

[a] Commissariat à l'Énergie Atomique et Aux Énergies Alternatives, CEA, DEN/DANS/DM2S/STMF/LMEC, CEA Cadarache, Bât. 238, F-13108 St Paul-lez-Durance, France
[b] Commissariat á l'Énergie Atomique et Aux Énergies Alternatives, CEA, DEN/DANS/DM2S/STMF/LMSF, CEA Grenoble, Bât 1005, 17 rue des Martyrs, F-38054 Grenoble Cedex 9, France

ARTICLE INFO

Article history:
Received 28 August 2012
Received in revised form 20 September 2013
Accepted 31 October 2013
Available online 20 November 2013

Keywords:
Immersed boundaries
Penalized direct forcing
Projection scheme
Cartesian grids
Interpolation scheme

ABSTRACT

In this paper, we propose a second order penalized direct forcing method to deal with fluid–structure interaction problems involving complex static or time-varying geometries. As this work constitutes a first step toward more complicated problems, our developments are restricted to Dirichlet boundary condition in purely hydraulic context. The proposed method belongs to the class of immersed boundary techniques and consists in immersing the physical domain in a Cartesian fictitious one of simpler geometry on fixed grids. A penalized forcing term is added to the momentum equation to take the boundary conditions around/inside the obstacles into account. This approach avoids the tedious task of re-meshing and allows us to use fast and accurate numerical schemes. In contrary, as the immersed boundary is described by a set of Lagrangian points that does not generally coincide with those of the Eulerian grid, numerical procedures are required to reconstruct the velocity field near the immersed boundary. Here, we develop a second order linear interpolation scheme and we compare it to a simpler model of order one. As far as the governing equations are concerned, we use a particular fractional-step method in which the penalized forcing term is distributed both in prediction and correction equations. The accuracy of the proposed method is assessed through 2-D numerical experiments involving static and rotating solids. We show in particular that the numerical rate of convergence of our method is quasi-quadratic.

1. Introduction

Fluid flow with heat and mass transfer around complex stationary or moving geometries (solid or flexible) appears in a large number of situations of practical interest including biological fluid mechanics (blood flow in human heart for instance) or in life-science context (the fish-like swimming *e.g.*). Fluid–structure interaction problems are also of importance in many engineering applications, as for example, to design industrial heat exchangers, aerospace vehicles or in nuclear safety context. In this latter case, the vitrification process for the radioactive waste storage is an example. In this process, a viscous multiphase multicomponent flow at high temperature (gas bubbles and molten glass incorporating the ultimate waste) interacts with both static (*e.g.* the vessel structure, the apparatus of measurement, etc.) and moving (*e.g.* the mechanical stirrer) bodies of more or less complex geometries.

The numerical treatment of these kinds of problem appears to be a challenging task because of time-varying geometries, often combined with complex flow regimes. To tackle numerically these complex problems, the well-known body-fitted approach is usually followed. Such an approach consists in discretizing the governing equations on a non-structured mesh for which the boundaries of the computational domain lie on those of the physical domain. Thereby, boundary conditions are directly (and so, exactly) imposed on the physical domain boundary. However, the main drawback of the body-fitted like techniques lies in their lack of ability to handle complex industrial problems involving moving bodies which require the development of specific numerical schemes to deal with the difficult issue of re-gridding.

Another approach consists in using non-boundary conforming techniques in which the physical domain is immersed in a fixed fictitious one of simpler geometry on a Cartesian grid. Such techniques allow us to use efficient, fast and accurate numerical methods avoiding the tedious task of the re-meshing caused by time-varying geometries. In contrast, as the immersed boundaries are described by a set of Lagrangian points (or the zero of a level-set function) that do not generally coincide with those of the Eulerian grid, numerical methods have to account for the immersed boundary conditions at their right places. The non-boundary conforming techniques proposed in the literature may be classified into two categories.

The first category, including for instance Cartesian methods (*e.g.* [1,2]), the Immersed Interface Method (IIM, [3]) or the Jump Embedded Boundary Condition method (JEBC, [4]), mimics the presence of embedded geometries by modifying the numerical

* Corresponding author.
E-mail address: michel.belliard@cea.fr (M. Belliard).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.10.044

scheme in the immediate vicinity of the immersed boundary or interface. The two latter methods introduce jump conditions across the interface in the solve of the partial differential equations. Such an approach leads to a sharp representation of the immersed interface but, for the Cartesian method, extending it to three-dimensional problems may appear to be a challenging task, particularly regarding the coding logistic.

In the second category (rather than locally modifying the numerical scheme) a supplementary term, referred to as the forcing term, is added to the governing equations. This class of non-boundary conforming techniques dates back to Peskin's works in which an Immersed Boundary Method (IBM) has been developed to numerically simulate blood flows in a human heart [5]. In this case, the immersed boundaries correspond to muscular heart walls and extra forces acting in these boundaries are modeled by a vectorial forcing term added to the continuous Navier–Stokes equations. A Lagrangian coordinate system is employed to track the interface and to calculate the vectorial forcing term. The IB method has been successfully applied to problems with elastic geometries but, in the rigid limit, it generally leads to very stiff problems. Moreover, in order to ensure the stability of the numerical scheme, the forcing term based on a Dirac delta function must be smeared over a stencil of few Cartesian nodes. Following the ideas introduced by Peskin, several IB-like methods with different forcing terms (or forcing strategies) have been proposed in the literature. In [6], Goldstein et al. propose the Feedback Forcing (FF) method in which the forcing term can be viewed as a force density that brings the fluid velocity to zero near the immersed boundary. Similarly to what is done in [5], the numerical scheme used in [6] requires a spreading of the forcing term over the interface. Moreover, the FF method suffers from the fact that the forcing term highly depends on flow properties. Whether the Peskin's IB method or the FF method, their application to flows at high Reynolds number is limited by the spreading of the forcing term over the immersed boundary. In this case, local mesh refinement techniques can be a solution [7]. An alternative approach to the aforementioned techniques, referred to as Direct Forcing (DF) method, has been proposed by Mohd-Yusof [8] and then adapted by Fadlun et al. [9]. This immersed boundary technique consists in directly applying the desired boundary conditions on Cartesian nodes close to the interface leading to a quasi sharp representation of the interface (through one cell layer). In that sense, using the terminology employed by Gilmanov et al. in [10], the DF method may be referred to as a Hybrid Cartesian/Immersed Boundary (HCIB) approach and may be conceptually related to the IIM. Moreover, one of the interest of the DF method is that the forcing term can be easily computed and it does not depend on the flow properties. Therefore, the stability of the numerical scheme is not affected. However, the accuracy of the DF method is partially dependent on the numerical scheme because the calculation of the forcing term is in particular based on the discretized form of the governing equations. Since its development by Mohd-Yusof [8], the DF method has gained in popularity and has been successfully applied to various fluid–structure interaction problems (*e.g.* [10–15]) or turbulent flow simulations (*e.g.* [16,17]) using mesh refinement or mesh stretching techniques. It is also worth to mention the immersed boundary method of Pinelli et al. [18], that has roots in both IBM and DF methods, and which is suitable for general grid systems including curvilinear ones. More recently, Belliard and Fournier [19] have proposed a variant of HCIB techniques, called Penalized Direct Forcing (PDF) method, that combines both the basic features of the DF method and those of L^2-penalty methods (*e.g.* [20]). Links can be found with the works of Sarthou et al. [21] and those of Bergmann and Iollo [22]. As for the DF method, the unknowns are locally enforced on the grid nodes nearest the immersed interface. However, the PDF method appears to be a more versatile approach than the DF method because the forcing term expresses as a L^2-penalty term that is independent on the discrete governing equations.

In the present paper, after introducing the discretization of the Navier–Stokes governing equations in the Section 2, the PDF algorithm is detailed in the Section 3, including a specific treatment of the pressure near the immersed boundaries. The PDF method itself is presented in Section 3.1. Interesting for practical purposes, the non-boundary conforming approaches are often coupled with fractional-step schemes (*e.g.* [23,24]) but, as emphasized by Ikeno and Kajishima [25] or Taira and Colonius [26], most of them take account of the forcing term only in the prediction equation leading to inconsistent schemes. Here, an original fractional-step scheme leading to a consistent (in the sense of [25]) PDF method is developed and presented in Section 3.2. Homogeneous Neumann IBCs for the pressure are recovered through a particular treatment of the pressure equation coefficients near the immersed interface.

Whatever the non-boundary conforming method involved, an important issue concerns the reconstruction of the velocity field close to the immersed boundary and the accuracy of the numerical method developed for this purpose. These points have received a particular attention in the literature with, most of the time, the development of interpolation schemes. Most of the classical approaches consist in interpolating or extrapolating the velocity field in a preferred direction (*e.g.* [10,12]). In this work, we have developed an original robust interpolation scheme, second-order accurate in space, that is not guided by particular direction. It relies mainly on an averaged reconstruction of the velocity gradient near the IB and on an approximate projection operator onto the IB. Without loss of generality, we restrict our presentation to Dirichlet's IBCs for the velocity.[1] This is the object of Section 4.

Finally, in Section 5, some 2-D numerical experiments are performed for steady and unsteady incompressible laminar flows at very moderate Reynolds number (up to 100) around/between static and rotating solids to assess the validity, accuracy and the ability of the proposed method for both uniform and analytically velocity prescribed at IBs. We show in particular that the numerical rate of convergence is (quasi-) quadratic for all studied cases. To highlight the ability of our method to deal with 3-D moving complex geometries, we also present an illustration of flows induced by a stirrer.

2. Governing equations and numerical method

The section is devoted to the numerical method. It is structured in two parts. The first part focuses on the governing equations whereas the numerical scheme is the object of the second part.

2.1. Governing equations

The governing equations used to describe unsteady incompressible flows are given by:

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \otimes \mathbf{u}) + \nabla P - \nu \nabla^2 \mathbf{u} = \mathbf{f} \quad \text{in } \Omega \tag{1a}$$

$$\nabla \cdot \mathbf{u} = 0 \quad \text{in } \Omega \tag{1b}$$

$$\mathbf{u} = \mathbf{u}_D \quad \text{on } \partial\Omega \text{ and } \mathbf{u}(t_0) \text{ given in } \Omega \tag{1c}$$

where Ω denotes the computational domain, $\partial\Omega$ its boundary, $\mathbf{u}$ the solenoidal velocity and ν the kinematic viscosity. Here-above, P is the total pressure defined by:

$$\rho \nabla P = \nabla p - \rho \mathbf{g} \tag{2}$$

[1] Neumann IBCs can be also considered by interpolations involving the prescribed flux at the boundary and the velocities of the surrounding flow.