Commun. Comput. Phys.
doi: 10.4208/cicp.220313.111013a

Vol. **15**, No. 5, pp. 1266-1290
May 2014

An Accurate Cartesian Method for Incompressible Flows with Moving Boundaries

M. Bergmann*, J. Hovnanian and A. Iollo

Inria, F-33400 Talence, France, Univ. Bordeaux, IMB, UMR 5251, F-33400 Talence, France, CNRS, IMB, UMR 5251, F-33400 Talence, France.

Received 22 March 2013; Accepted (in revised version) 11 October 2013

Communicated by Boo-Cheong Khoo

Available online 14 February 2014

Abstract. An accurate cartesian method is devised to simulate incompressible viscous flows past an arbitrary moving body. The Navier-Stokes equations are spatially discretized onto a fixed Cartesian mesh. The body is taken into account via the ghost-cell method and the so-called penalty method, resulting in second-order accuracy in velocity. The accuracy and the efficiency of the solver are tested through two-dimensional reference simulations. To show the versatility of this scheme we simulate a three-dimensional self propelled jellyfish prototype.

AMS subject classifications: 68Uxx, 76

Key words: Viscous incompressible flow, immersed boundary method, penalty method, cartesian grid, self-propelled jellyfish.

1 Introduction

Over the last decades, many works have been developed in order to precisely describe the interactions between fluids and structures, especially when the solid presents complex boundaries. These studies can be divided in two categories. The first category is represented by Arbitrary Lagrangian Eulerian methods [10]. These methods are accurate but are hard to set up and complicated to implement, moreover when moving and/or deforming obstacles are considered. In addition, a dynamic mesh partitioner is necessary when one deals with parallel computations. The second category is represented by interface methods that are usually more versatile. Among them, there are the Ghost Fluid method [12] or the Immersed Boundary method [24]. The current work focuses on the immersed boundary method on a cartesian grid.

*Corresponding author. *Email addresses:* `michel.bergmann@inria.fr` (M. Bergmann), `jesshovna@msn.com` (J. Hovnanian), `angelo.iollo@math.u-bordeaux1.fr` (A. Iollo)

The immersed boundary method has initially been introduced by Peskin in 1972 [29]. In his original version, the interactions between fluid and solid are described through a forcing term consisting in a Dirac function located at the interface. This term is added to the Navier-Stokes equations before discretization and thus is completely independent of the spacial discretization. This approach is defined as a *continuous forcing* method and was used by Peskin to simulate blood flow in a beating heart but has also been implemented to simulate multiphase flow [36, 38] and solidification [40]. In another class of immersed boundary methods one modifies the space discretization of the Navier-Stokes equations near the interface. They are called *discrete forcing* methods. This method was used and developed by Liu et al. [19], Marella et al. [21], Mittal et al. [11,23], Li et al. [17], among others. Despite being strongly dependent on the space discretization, this second category has the advantage of being sharp, as opposed to the first one.

The Ghost Cell method is inspired by this last approach since ghost-cell values are introduced based on an interpolation of neighboring fluid cells, such that the boundary conditions are satisfied on the immersed interface.

In this paper, the resolution of the Navier-Stokes equations follows a classical predictor-corrector scheme [5, 37] with second-order spatial accuracy near the immersed boundary. Other schemes have recently been proposed to achieve higher-order accuracy for simulating flows with moving boundaries on cartesian meshes, see for example [11, 18, 21, 23]. In those works, the main idea is to improve accuracy by an efficient and simple reconstruction of the solution near the immersed boundary. The schemes then usually differ near the immersed boundary in the specific solution reconstruction and in the treatment of mesh nodes that present geometric ambiguities with respect to the discretization stencil. In the same spirit, here we propose a local reconstruction of the solution near the immersed boundary that is based on the geometric information delivered by the distance function [26, 32, 33]. Moreover, as explained in the following, a penalty method [1] is employed in the prediction step, allowing a consistent and efficient solution of the pressure equation on the whole domain. Also, as it is shown hereafter, the penalty approach automatically solves the problem of fixing appropriate physically meaningful conditions for the "fresh" nodes crossing the immersed boundary when the border is moving.

The organization of this paper is as follows: the Section 2 is devoted to the description of the numerical methodology including the scheme used at the immersed boundary. In Section 3, we present validations through computed results for several canonical tests and establish the accuracy of the scheme. The present method is then used to simulate the flow generated by a self-propelled jellyfish-like swimmer. Finally, the Section 4 is dedicated to conclusions.

2 Discretization of the governing equations

We call Ω_f the fluid domain, surrounding a body called Ω_s. The entire domain is