Available online at www.sciencedirect.com
SCIENCE DIRECT®

ELSEVIER

Journal of Computational Physics 215 (2006) 12–40

JOURNAL OF COMPUTATIONAL PHYSICS

www.elsevier.com/locate/jcp

An embedded-boundary formulation for large-eddy simulation of turbulent flows interacting with moving boundaries

Jianming Yang, Elias Balaras *

Department of Mechanical Engineering, University of Maryland, College Park, MD 20742, USA

Received 31 October 2004; received in revised form 7 June 2005; accepted 19 October 2005
Available online 19 December 2005

Abstract

A non-boundary-conforming formulation for simulating complex turbulent flows with dynamically moving boundaries on fixed Cartesian grids is proposed. The underlying finite-difference solver for the filtered incompressible Navier–Stokes equations is based on a second-order fractional step method on a staggered grid. To satisfy the boundary conditions on an arbitrary immersed interface, the velocity field at the grid points near the interface is reconstructed using momentum forcing without smearing the sharp interface. The concept of field-extension is also introduced to treat the points emerging from a moving solid body to the fluid. Laminar flow cases and large-eddy simulations (LES) are presented to demonstrate the formal accuracy and range of applicability of the method. In particular, simulations of laminar flow induced by the harmonic in-line oscillation of a circular cylinder in quiescent fluid, and from a transversely oscillating cylinder in a free-stream are presented and compared to reference simulations and experiments. LES of turbulent flow over a traveling wavy wall and transitional flow through a bileaflet prosthetic heart valve are also shown. All results are in very good agreement with reference results in the literature.

Keywords: Large-eddy simulation; Immersed-boundary method; Fluid–structure interaction; Moving boundaries; Cartesian grid; Finite-difference method

1. Introduction

Recently there has been renewed interest in the development of non-boundary-conforming methodologies for the solution of the Navier–Stokes equations. In such methods the requirement that the grid conforms to a solid boundary is relaxed, and the effect of a complex object on the flow is introduced through proper treatment of the solution variables at the grid cells in the vicinity the body. The basic advantage of these formulations is the simplification of grid generation, especially in cases of moving boundaries where the need for regeneration or deformation of the grid is eliminated. In addition, efficient Cartesian solvers can be directly applied to complex flow problems. Both the above features are particularly attractive for Direct Numerical

* Corresponding author. Fax: +1 301 314 9477.
E-mail address: balaras@eng.umd.edu (E. Balaras).

0021-9991/$ - see front matter
doi:10.1016/j.jcp.2005.10.035

Simulations (DNS) or large-eddy simulations (LES) of turbulent and transitional flows, where the use of highly efficient, energy conserving solvers is imperative for accurate computations. It is therefore conceivable that successful integration of non-boundary-conforming strategies with robust Cartesian or cylindrical coordinate solvers developed for DNS/LES will open a wide new area of applications for these tools. Example applications include a variety of low and moderate Reynolds number turbulent flow problems from engineering, biology, and medicine, where fluid/structure interactions are central to the dynamics of the flow.

Over the past decades a variety of non-boundary-conforming methods with various degrees of accuracy and complexity have been proposed. The so-called immersed-boundary formulation pioneered by Peskin [19] represents a family of methods where a set of body forces is used to represent the effect of an object to the flow. Initially the method was used to study fluid–structure interaction problems in the cardiovascular circulation [20,21]. In these computations the vascular boundary was modeled as a set of elements linked by springs. As a result the forces required to enforce boundary conditions could be evaluated in a straightforward manner (i.e. Hooke's law). In the case of rigid immersed bodies, however, the corresponding forces are not known a priori and must be calculated by some feedback algorithm. Lai and Peskin [15] suggested a formulation where the body is allowed to move a little – rather than being fixed – by connecting it to a very stiff spring. They tested this approach for the flow around a cylinder with satisfactory results, although the specification of the stiffness constant is somehow ad hoc. Goldstein et al. [7] introduced an alternative approach where a feedback-forcing scheme is used to asymptotically enforce the desired boundary conditions on a solid boundary. Application of the method to three-dimensional computations of turbulent flow in plane and ribbed channels yield results in good agreement with reference data [8,9]. An advantage of immersed-boundary formulations is their straightforward implementation in existing solvers (modifications are confined to the RHS of the equations of motion). On the other hand, the need for a smooth transition between the fluid and the solid body spreads the forcing function over several grid cells and introduces some blurring between the two regions. This feature can decrease the order of accuracy of the scheme near the body or increase the resolution requirements making their use in turbulent flows problematic.

Another class of methods which does not suffer from the ''blurring'' mentioned above are the so-called Cartesian or cut-cell formulations. In this case the solid boundary is tracked as a sharp interface and the grid cells at the body interface are modified according to their intersections with the underlying Cartesian grid. The discrete operators at these cells are then modified to reflect the desired boundary conditions. Successful applications of such methods in two-dimensional flows with stationary boundaries can be found in [33,3,23,27,31]. However, due the large number of possible intersections between the grid and the boundary a variety of interface-cells is generated leading to an equally large number of special treatments. Also in complex configurations the unavoidable generation of irregularly shaped cells with very small size can have an adverse impact on the conservation and stability properties of the solver. Recently Ye et al. [31] suggested a cell merging scheme to address this problem. This formulation was also extended to treat moving boundaries with good results for a variety of two-dimensional problems [29]. The extension of the methodology in complex three-dimensional configurations remains to be investigated.

Recently, Fadlun et al. [6] introduced an embedded-boundary method, which shares a number of features with both approaches discussed above. As in immersed-boundary formulations the method still utilizes a force-field to enforce boundary conditions. In this case, however, the forces are not specified in the continuous space by means of some physical arguments, but rather in the discrete space by directly requiring the solution to respect the desired boundary conditions. This process is equivalent to a local reconstruction of the solution near the interface and enforces the desired boundary conditions ''exactly'', as in cut-cell formulations. The encouraging results reported in [30,6] together with the straightforward implementation of the method in existing Navier–Stokes solvers, motivated a number of recent studies, where alternative embedded-boundary formulations based on the principles outlined above have been proposed. The main difference between them is the way the solution is reconstructed near the interface. In [6,1], for example, the solution is reconstructed at the fluid nodes closest to the immersed boundary (fluid points with at least one neighbor in the solid phase). In the former study an one-dimensional interpolation scheme along an arbitrary grid line is used for this purpose, while in the latter the reconstruction is performed along the well-defined line normal to the interface. In [14,16], or [26] on the other hand, the solution is reconstructed at 'ghost-cells', which are points inside the solid phase with at least one neighbor in the fluid phase. Both the above strategies have the velocity boundary