Computers and Fluids 170 (2018) 77–92

Contents lists available at ScienceDirect

Computers and Fluids

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

Immersed boundary method for viscous compressible flows around moving bodies

M. Ehsan Khalili[a,*], Martin Larsson[b], Bernhard Müller[a]

[a] Department of Energy and Process Engineering, Norwegian University of Science and Technology (NTNU), Kolbjørn Hejes vei 2, Trondheim NO-7491, Norway
[b] Sportradar AS, Ferjemannsveien 10, Trondheim NO-7014, Norway

ARTICLE INFO

Article history:
Received 27 December 2017
Revised 13 April 2018
Accepted 27 April 2018
Available online 30 April 2018

Keywords:
Immersed boundary method
Sharp interface method
High order finite difference method
Compressible Navier–Stokes equations
Moving boundaries

ABSTRACT

A ghost-point immersed boundary method for the compressible Navier–Stokes equations with moving boundaries on fixed Cartesian grids is devised by employing high order summation-by-parts (SBP) difference operators. The immersed boundaries are treated as sharp interfaces by enforcing the solid wall boundary conditions via flow variables at ghost points using bilinearly interpolated flow variables at mirror points. Simulations for compressible viscous flows induced by a transversely oscillating cylinder in a free-stream and a harmonic in-line oscillating cylinder in an initially quiescent fluid are presented and compared with experiments and incompressible fluid flow simulations obtained with body-conforming grid methods.

1. Introduction

Numerous engineering, biological and biomedical systems involve moving boundaries and fluid structure interaction. Conventional computational approaches for moving boundaries are usually based on body-conforming grids such as the arbitrary Lagrangian–Eulerian (ALE) formulation [1–3]. In this method, a Lagrangian mesh following the material points is employed to discretize the solid body, and a moving mesh is used to discretize the flow domain and to conform to the instantaneous configuration of the solid body. Although the boundary conditions of the flow can be easily imposed at the solid body in body-conforming methods, an algorithm is required to move the mesh points in the fluid flow domain along with solid boundary movement or deformation.

On the other hand, there has recently been a growing interest to develop non-body conforming methodologies to solve these types of multidisciplinary problems. In such methods, the grid is not required to conform to the solid body and moving boundaries. The major advantage of these methods is that they simplify the grid generation process, particularly in cases of moving boundaries where the requirement of re-meshing is eliminated. There are several approaches in computational fluid dynamics (CFD) to treat complex or moving boundary problems based on non-body fitted Cartesian grids. The most notable among them is the immersed boundary method (IBM). According to Mittal and Iaccarino [4], IB methods can be broadly classified into two categories, namely a continuous forcing approach and a sharp interface method (discrete forcing) based on a procedure of imposing boundary conditions [4]. In the first category which was originated by Peskin [5,6], a forcing function is included in the momentum equation and then applied to the entire domain. Peskin [5] introduced the term of "immersed boundary method" and developed the approach to simulate blood flow in the cardiovascular system to deal with elastic boundaries. Goldstein et al. [7] developed the continuous forcing approach to model the effect of rigid bodies by using feedback forcing. The significant advantage of continuous forcing approaches is that they are independent of the underlying spatial discretization. However, the major drawbacks of these types of IBM are that they may induce spurious oscillations, and exhibit numerical instability issues, particularly for unsteady flows at high Reynolds numbers due to the inherent stiffness of the forcing terms [7,8].

In the direct forcing approach proposed by Mohd-Yusof [9], the governing equations are discretized on a Cartesian grid without computing any forcing term directly. In this approach, the effect of the body force is calculated by determining the difference between the mirrored velocity at internal points inside the body and the velocity at external points outside the body to enforce the tangential velocity at the immersed boundary. Fadlun et al. [10] further implemented linear interpolations for the reconstruction of the velocity at the grid point near the solid body boundary so that the interpolation direction can be chosen arbitrarily. Balaras [11] modified

* Corresponding author.
E-mail address: mohammadtaghi.khalili@ntnu.no (M. Ehsan Khalili).

https://doi.org/10.1016/j.compfluid.2018.04.033
0045-7930/

the approach by using the reconstruction scheme and interpolation along the direction normal to the body surface. In the work by Gilmanov et al. [12], the restriction of a linear interpolation scheme for the velocity along the local normal direction was extended to arbitrarily complex, three-dimensional geometries by discretizing the body surface using triangular panels. They also extended their immersed boundary methods to moving-body simulations at different Reynolds numbers [13,14]. Gilmanov et al. [13] considered arbitrarily complex moving bodies by employing a second order hybrid staggered/non-staggered grid approach. Yang and Balaras [14] introduced the concept of field-extension to treat the points emerging from a moving solid body to the fluid. These methods generally rely on reconstructions of the solution at the so-called forcing nodes (fluid points with at least one neighboring point inside the solid body).

Another direct forcing approach which reconstructs the solution at the ghost cells was introduced by Tseng and Ferziger [15]. Ghost cells are defined as cells within the solid body having at least one neighboring cell inside the fluid domain [16]. Mittal et al. [16] have shown the large potential of the ghost cell approach to deal with highly complex geometries as well as moving and deforming bodies. The concept of the image point which is the mirror of the ghost-cell point along the normal direction to the body surface was first introduced in the works [16,17]. They constructed interpolation operators in the normal direction to the IB in order to simplify the implementation of Neumann boundary conditions at the IB [16,17].

Nonetheless, most immersed boundary methods are designed for incompressible flows [4]. Investigations of viscous compressible flows particularly in interaction with moving bodies are still scarce. Only few IB methods for viscous compressible flows and acoustic wave propagation problems have been developed [18]. Due to the different mathematical characteristics of the Navier–Stokes equations for compressible and incompressible flows, there are differences in the implementation of the boundary conditions between these two types of equations as well as in the spatial discretization schemes employed [18,19].

Global and local conservation can be obtained by the Cartesian cut-cell method for immersed boundary problems [20,21]. In the cut-cell method, a finite volume scheme is designed to represent the conservation laws also for cells cut by the immersed boundary. Since this method resolves the forces at the immersed boundary and directly discretizes the conservation laws for mass, momentum, and energy, this method is especially attractive for moving boundary problems. However, the wide range of possibilities of geometrical shapes for cut-cells (complex polyhedral cells) causes difficulties in extending the method to 3D and implementing it for arbitrarily complex geometries.

In this study, the ghost point IB approach has been adopted for treating moving immersed boundaries using a high order finite difference method based on summation-by-parts (SBP) operators to provide an accurate and efficient approach for studying low Mach number compressible viscous flows. The main focus in our study is the presentation, verification and validation of our high order IBM. Our method allows to simulate not only subsonic viscous flow around moving bodies, but also acoustic wave propagation. The proposed approach is verified and validated for two dimensional flows over in-line and transversely oscillating circular cylinders.

The paper is organized as follows. In Section 2, the model for fluid flow is presented. In Section 3, the numerical method is described. The immersed boundary formulation for moving bodies, the treatment of newly emerged fluid points and the implementation of the boundary conditions are explained. In Section 4, results are provided and compared with numerical and experimental data available in the literature. Conclusions are stated in Section 5.

2. Compressible Navier–Stokes equations

In the present study, the 2D compressible Navier–Stokes equations in perturbation form are solved. The perturbation formulation is employed to minimize cancellation errors when discretizing the Navier–Stokes equations for compressible low Mach number flow [22,23]. The conservative form of the 2D compressible Navier–Stokes equations in perturbation formulation can be written as

$$\mathbf{U}'_t + \mathbf{F}^{c'}{}_x + \mathbf{G}^{c'}{}_y = \mathbf{F}^{v'}{}_x + \mathbf{G}^{v'}{}_y, \tag{1}$$

where $\mathbf{U}' = \mathbf{U} - \mathbf{U_0}$ is the vector of conservative perturbation variables with $\mathbf{U} = (\rho, \rho u, \rho v, \rho E)^T$ the vector of the conservative variables and $\mathbf{U_0} = (\rho_0, 0, 0, (\rho E)_0)^T$ the stagnation values.

The conservative perturbation variables $\mathbf{U}'$ and the inviscid ($\mathbf{F}^{c'}$, $\mathbf{G}^{c'}$) and viscous perturbation flux vectors ($\mathbf{F}^{v'}$, $\mathbf{G}^{v'}$) are defined by $\mathbf{F}^{c'} = \mathbf{F}^c(\mathbf{U}) - \mathbf{F}^c(\mathbf{U_0})$, etc., according to

$$\mathbf{U}' = \begin{pmatrix} \rho' \\ (\rho u)' \\ (\rho v)' \\ (\rho E)' \end{pmatrix}, \quad \mathbf{F}^{c'} = \begin{pmatrix} (\rho u)' \\ (\rho u)'u' + p' \\ (\rho v)'u' \\ ((\rho H)_0 + (\rho H)')u' \end{pmatrix},$$

$$\mathbf{G}^{c'} = \begin{pmatrix} (\rho v)' \\ (\rho u)'v' \\ (\rho v)'v' + p' \\ ((\rho H)_0 + (\rho H)')v' \end{pmatrix}, \quad \mathbf{F}^{v'} = \begin{pmatrix} 0 \\ \tau'_{xx} \\ \tau'_{xy} \\ u'\tau'_{xx} + v'\tau'_{xy} + \kappa T'_x \end{pmatrix},$$

$$\mathbf{G}^{v'} = \begin{pmatrix} 0 \\ \tau'_{yx} \\ \tau'_{yy} \\ u'\tau'_{yx} + v'\tau'_{yy} + \kappa T'_y \end{pmatrix},$$

where t is the physical time and x and y are the Cartesian coordinates. ρ denotes density, u and v the x- and y-direction velocity components, E the specific total energy, T the temperature and κ the heat conduction coefficient calculated from the constant Prandtl number $\mathrm{Pr} = 0.72$. ρ_0, $(\rho E)_0$ and $(\rho H)_0$ denote the stagnation values of density, total energy density and total enthalpy density, respectively. The perturbation variables are given as follows

$$\rho' = \rho - \rho_0, \quad (\rho\mathbf{u})' = (\rho\mathbf{u}), \quad (\rho E)' = \rho E - (\rho E)_0,$$
$$(\rho H)' = (\rho E)' + p', \mathbf{u}' = \frac{(\rho\mathbf{u})'}{\rho_0 + \rho'},$$
$$\tau' = \mu(\nabla\mathbf{u}' + (\nabla\mathbf{u}')^T) - \frac{2}{3}\mu(\nabla\cdot\mathbf{u}')\mathbf{I},$$
$$T' = \frac{p'/R - \rho' T_0}{\rho_0 + \rho'}.$$

Here, R is the specific gas constant and μ is the viscosity which is determined from the Sutherland law $\frac{\mu}{\mu_0} = (\frac{T}{T_0})^{1.5}[(1 + S_c)/(\frac{T}{T_0} + S_c)]$ with the non-dimensional Sutherland constant $S_c = \frac{110}{301.75}$.

Since perfect gas is considered, the pressure perturbation can be related to the conservative perturbation variables $p' = (\gamma - 1)[(\rho E)' - \frac{1}{2}((\rho\mathbf{u}'\cdot\mathbf{u}'))]$, where the ratio of specific heats $\gamma = c_p/c_v = 1.4$ for air.

The flow variables are non-dimensionalized with ρ_0, stagnation speed of sound c_0 and $\rho_0 c_0^2$ as reference values. In order to generalize the geometry for non-uniform Cartesian grids, the equations of motion are transformed from the physical domain (x, y) to the computational domain (ξ, η) by a mapping,

$$\xi = \xi(x, y),$$
$$\eta = \eta(x, y). \tag{2}$$

Thus, the transformed 2D compressible Navier–Stokes equations in perturbation form are expressed as:

$$\hat{\mathbf{U}}'_t + \hat{\mathbf{F}}'_\xi + \hat{\mathbf{G}}'_\eta = 0, \tag{3}$$