Computers and Fluids 136 (2016) 272–284

Contents lists available at ScienceDirect

Computers and Fluids

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

A comparative study of Brinkman penalization and direct-forcing immersed boundary methods for compressible viscous flows

A. Piquet [a,b], O. Roussel [a,c], A. Hadjadj [a,*]

[a] INSA and University of Normandy, Avenue de l'Université, 76801 Saint-Étienne-du-Rouvray, France
[b] Department of Mechanical Engineering, Imperial College London, UK
[c] Department of Physics, Umea University, 90187 Umea, Sweden

ARTICLE INFO

Article history:
Received 21 November 2015
Revised 11 May 2016
Accepted 1 June 2016
Available online 9 June 2016

Keywords:
Brinkmann penalization
Direct-forcing
Immersed boundary method
Shock waves
Supersonic channel flow

ABSTRACT

This paper deals with the comparison between two methods to treat immersed boundary conditions: on the one hand, the Brinkman penalization method (BPM); on the other hand, the direct-forcing method (DFM). The penalty method treats the solid as a porous medium with a very low permeability. It provides a simple and efficient approach for solving the Navier–Stokes equations in complex geometries with fixed boundaries or in the presence of moving objects. A new approach for the penalty-operator integration is proposed, based on a Strang splitting between the penalization terms and the convection-diffusion terms. Doing so, the penalization term can be computed exactly. The momentum term can then be computed first and then introduced into the continuity equation in an implicit manner. The direct-forcing method however uses ghost-cells to reconstruct the values inside the solid boundaries by projection of the image points from the interface. This method is comparatively hard to implement in 3D cases and for moving boundaries. In the present paper, the performance of both methods is assessed through a variety of test problems. The application concerns the unsteady transonic and supersonic fluid flows. Examples include a normal shock reflection off a solid wall, transonic shock/boundary layer in a viscous shock tube, supersonic shock/cylinder interaction, and supersonic turbulent channel flow. The obtained results are validated against either analytical or reference solutions. The numerical comparison shows that, with sufficient mesh resolution, the BPM and the DFM methods yield qualitatively similar results. In all considered cases, the BPM is found to be a suitable and a possibly competitive method for viscous-IBM in terms of predictive performance, accuracy and computational cost. However, despite its simplicity, the method suffers from a lack of regularity in the very near-wall pressure fluctuations, especially for the turbulent case. This is attributed to the fact that the method requires no specific pressure condition at the fluid/solid interface.

1. Introduction

Numerical simulations of viscous flows around solid obstacles or within boundaries of arbitrary shape are of crucial interest in many engineering applications. Up to now, two main approaches have been followed to deal with complex geometries: body-fitted grid methods and immersed boundary methods. Body-fitted methods rely on structured or unstructured grids that are generated to fit with complex boundaries. Therefore, to obtain accurate solutions for complex geometries, it is required to refine the mesh near the boundary-layer region. Nevertheless, to built a body-fitted grid, an expensive grid generation process has to be followed. Moreover, even with simple geometries, it is still difficult to generate a mesh of good quality. Finally, for a given numerical scheme, the order of accuracy on structured or unstructured grids is always lower than on Cartesian grids. An alternative approach is to perform simulations on non-body conformal Cartesian grids and to impose immersed boundary conditions on the fluid. The main advantages of this approach are its easy implementation and the possibility to treat moving boundaries in a simpler manner.

Since Peskin's pioneering work [31], various immersed boundary techniques have been developed, mostly for incompressible flows. They can be decomposed into two categories: the methods that introduce fictitious terms in the governing equations and those which locally modify the structure of the background grid. In the first category, Peskin [31] modeled immersed boundaries as elastic media that exert localized forces on the fluids and hence modify the momentum equation. Various extensions to rigid body

* Corresponding author.
E-mail address: hadjadj@coria.fr (A. Hadjadj).

http://dx.doi.org/10.1016/j.compfluid.2016.06.001

problems have been proposed (Goldstein et al [16], Saiki and Biringen [34]). However, these methods used explicit-time stepping for such problems, which are in fact stiff. Hence the computational time step is small, which gives severe restrictions to the method. Furthermore, there is no mathematical proof of convergence for these methods. In contrast to this approach, the volume penalization technique, proposed by Arquis and Caltagirone [3], models the solid body as a porous medium with very small permeability. A rigorous error estimation was proposed by Angot et al [2]. Angot [1] also proved that the solution of the penalized incompressible Navier–Stokes equations strongly converges towards the exact solution as the penalization parameter approaches zero. Several authors successfully applied this method to incompressible flows with fixed (Kevlahan and Ghidaglia [21]) as well as moving obstacles (Kadoch et al [20]). Liu and Vasilyev [24] applied the Brinkman penalization method to the compressible flow regime and Boiron et al [5] extended this approach to large Mach number flows. However, in these two papers, only isothermal walls have been considered. Another formulation of the volume penalization, which differs from the original idea of Angot and Caltagirone [3], is proposed by Brown-Dymkoski et al [6] that takes into account adiabatic walls and mixed boundary conditions.

In the second category, the direct-forcing immersed boundary method consists in using ghost cells and directly impose the boundary conditions on the immersed boundaries. This method has been introduced for incompressible flows by Mohd-Yousof [29]. The term of direct-forcing and the extension of the method to three-dimensional flows was proposed by Fadlun et al [12]. The bilinear interpolation (trilinear in 3D) of ghost points was introduced by Majumdar et al [27]. The method has then been successfully applied to turbulent flows [18], particulate flows [39], and fluid-structure interactions [35]. The extension to high-speed flows in the configuration of shock/obstacle interactions has been performed by Chaudhuri et al [8,10]. A general review of immersed boundary methods can be found in Mittal and Iaccarino [28].

The objective of the present paper is to compare the Brinkman penalization method, which is the most promising method of the first category, with the direct-forcing immersed boundary method, which is, to the best of our knowledge, the most efficient method of the second category. Generally, comparing different IBM algorithms is not straightforward as they differ in strategy, approach, computational complexity, and prediction ability. Moreover, such methods are strongly influenced by different selected parameters and test cases, and one wish to answer the following questions: Which approach is better? and do one of them have advantages over the other? To our best knowledge, no such comprehensive comparison of IBM methods applied to transonic / supersonic regimes is available in the literature so far. In the present paper, several test cases are performed to examine the behavior of fluid solid interaction, including shock wave propagation and reflection off a wall, shock/cylinder interaction and shock-free turbulence.

The paper is organized as follows: in Section 2, the numerical approach, including the governing equations, the penalized equations and the direct-forcing immersed boundary method, are presented. The obtained results are discussed in Section 3. Finally, conclusions and perspectives are drawn in Section 4.

2. Numerical method

2.1. Governing equations

Let $\Omega \in R^2$ be the computational domain containing N fixed regular obstacles ω_n, $n \in \{1, \ldots, N\}$, and let us set

$$\Omega_s = \bigcup_{n=1}^{N} \omega_n \text{ and } \Omega_f = \Omega \backslash \bar{\Omega}_s. \tag{1}$$

Here, $\bar{\Omega}_s$ denotes the closed region occupied by the solid bodies and Ω_f denotes the fluid domain.

For the fluid domain, we consider the compressible Navier-Stokes equations, together with appropriate boundary conditions on the solid bodies $\partial\omega_n$ and on the boundary of the computational domain $\partial\Omega$. The system of equations reads

$$\partial_t \mathbf{Q} + \nabla \cdot \mathbf{F}_I = \nabla \cdot \mathbf{F}_V \tag{2}$$

where $\mathbf{Q}$ denotes the vector of conservative variables

$$\mathbf{Q} = [\rho, \rho\mathbf{v}, \rho E]^T, \tag{3}$$

$\mathbf{F}_I$ denotes the inviscid flux tensor

$$\mathbf{F}_I = [\rho\mathbf{v}, \rho\mathbf{v} \otimes \mathbf{v} + p\mathbf{I}, (\rho E + p)\mathbf{v}]^T, \tag{4}$$

and $\mathbf{F}_V$ the viscous flux tensor

$$\mathbf{F}_V = [\mathbf{0}, \tau, \tau\mathbf{v} + \lambda\nabla T]^T. \tag{5}$$

Here ρ, $\mathbf{v} = [u, v, w]^T$, p, T and E denote the density, velocity, pressure, temperature and total energy per unit of mass of the fluid, respectively. λ is the thermal conductivity, $\mathbf{I}$ the identity matrix and

$$\tau = \mu\left[\nabla \otimes \mathbf{v} + (\nabla \otimes \mathbf{v})^T - \frac{2}{3}(\nabla \cdot \mathbf{v})\mathbf{I}\right] \tag{6}$$

where μ denotes the dynamic viscosity of the fluid, which follows Sutherland's law

$$\mu(T) = \mu_{ref}\left(\frac{T}{T_{ref}}\right)^{\frac{3}{2}} \frac{T_{ref} + T_S}{T + T_S} \tag{7}$$

with μ_{ref} and T_{ref} are the reference viscosity and temperature, respectively, and T_S is the Sutherland temperature. The system is closed by the equation of state for a calorically ideal gas

$$p = (\gamma - 1)\rho\left(E - \frac{\mathbf{v}^2}{2}\right) \tag{8}$$

with the isentropic exponent $\gamma = 1.4$ for air.

On the surface of each solid obstacle, the fluid velocity satisfies the no-slip condition

$$\mathbf{v}_{|\partial\omega_n} = \mathbf{v}_n. \tag{9}$$

We also consider that the wall temperature on each obstacle is fixed and, hence we impose Dirichlet boundary conditions for the temperature, i.e.

$$T_{|\partial\omega_n} = T_n. \tag{10}$$

It is worth noticing that the Neumann (or adiabatic) boundary condition can be easily implemented in the Direct-forcing method by applying the zero-gradient temperature condition at the wall $\partial T/\partial\vec{n} = 0$), i.e. Section 2.3.2. However, its implementation for the penalization method has lacked generality, especially for compressible flows. Recently, Brown-Dymkoski et al. [6] have proposed a Characteristic-Based Volume Penalization (CBVP) method that can address this question. However, in this paper, we will just focus on the isothermal condition for the sake of simplicity.

2.2. Space and time discretization

Space discretization is made through high-order finite differences. The inviscid fluxes are discretized using a fifth-order WENO scheme [19,25]. The principle relies on a convex combination of low-order polynomial reconstructions that yield a high-order resolution in smooth regions and keep the essentially non-oscillatory property near the discontinuities. Upwinding is made using a Roe scheme [33].

For sake of clarity, we present here the WENO scheme for a one dimensional scalar equation, considering the quantity φ. Extension