ANR project NORMA

Task 3: Geometrical strategies: IBM and mesh adaptation

The immersed boundary approach. A review.

Florian Miralles*, Bruno Koobus*

April 2021

*IMAG, Université de Montpellier, Montpellier, France
florian.miralles@umontpellier.fr, bruno.koobus@umontpellier.fr

Abstract

A review on immersed boundary (IB) methods is proposed in this document. This starts from Peskin's pioneering work on the application of the IB approach for the computation of the flow around heart valves [1] to recent works dealing with the computation of compressible turbulent flows using continuous or discrete forcing IB techniques [20, 56, 39]. Emphasis is placed on work dealing with IB methods applied to viscous compressible flows.

Keywords: Immersed boundary methods, continuous forcing approach, discrete forcing approach, penalization procedure, ghost-cell tecnhique, cut-cell technique, hybrid Cartesian/immersed boundary technique, Navier-Stokes equations, Cartesian mesh, unstructured mesh.

1 Introduction

In computational fluid dynamics (CFD), the classical numerical procedure to perform the simulation of fluid flows consists in discretizing the governing equations on body conforming grids in which nodes are coincident to the surface of an obstacle. Since the grids are constructed to precisely fit an obstacle, this technique allows for exact boundary conditions to be imposed. An alternative is the immersed boundary (IB) approach which allows the computation of a flow on a grid which does not conform to the solid boundary. In order to take into account solid body effects, this last approach introduces forcing in the constitutive

equations.
An important advantage of IB methods over classical body fitted methods is an easier grid generation since the grid does not have to fit the body. In IB methods, the grid complexity and quality are also not significantly affected by the complexity of the considered geometry. Furthermore, the IB approach can manage moving boundaries, possibly involving large deformations and displacements, in a simple and robust way without the need to generate a new grid at selected time steps. A disadvantage of this approach is that imposing the boundary conditions is not straightforward compared to body fitted methods. The accuracy and conservation properties of the numerical scheme are also not trivial. Another disadvantage of IB methods compared to body fitted methods is that the grid lines are not aligned with the body surface so that the accurate simulation of boundary layers is a complex and difficult task.
Several approaches have been developped by using the IB concept and differ in the discretization scheme used in the neighboring of the immersed surfaces. IB methods can be divided in two categories, namely the continuous forcing approach (or diffusive interface methods) and the discrete forcing approach (or sharp interface methods). In the first approach, a forcing function is introduced in the governing equations before the discretization to take into account the boundary conditions on the immersed surface. In the second approach, the discretization in the cells near the immersed boundary is modified, or the cells closed to the immersed surface are truncated, in order to directly impose the boundary conditions on the immersed surface.
A survey of some works carried out in the domain of IB methods is given in the following sections. The literature on the IB techniques is very rich and this review is not intended to be exhaustive. After having presented the main IB methods, we then focus on some works dealing with the simulation of viscous compressible flows using the IB approach. It is indeed this type of numerical applications that interests us more particularly within the framework of the ANR project NORMA.

2 IB methods: the main approaches

The various IB methods, which all discretize the governing equations on grids non conforming to the body, differ in the way in which the boundary conditions on the immersed boundary are taken into account.
Let us consider a viscous incompressible flow around a body which satisfies the following Navier-Stokes equations:

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \Delta \mathbf{u} \ \text{ in } \Omega_f \tag{1}$$

$$\nabla \cdot \mathbf{u} = 0 \ \text{ in } \Omega_f \tag{2}$$

$$\text{and } \mathbf{u} = \mathbf{u}_\Gamma \ \text{ on } \Gamma_b \tag{3}$$

where $\mathbf{u}$ is the fluid velocity, p the pressure, ρ the density and μ the viscosity. Ω_f and Γ_b denote respectively the fluid domain and the boundary of the solid which occupies the domain Ω_b.
In the IB approach, we disinguish two methodologies, namely the continuous forcing procedure and the discrete forcing procedure.
The continuous forcing technique consists in introducing in the governing equations an extra term, called forcing function and denoted here by $\mathbf{f} = (\mathbf{f}_m, f_p)$, in order to take into account

the boundary conditions that apply on the immersed surface :

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \Delta \mathbf{u} + \mathbf{f}_m \quad \text{in } \Omega_f \cup \Omega_b \tag{4}$$

$$\nabla \cdot \mathbf{u} = f_p \quad \text{in } \Omega_f \cup \Omega_b \tag{5}$$

The forcing terms used differ for an elastic boundary and a rigid boundary, with the introduction of models which try to mimic the effect of the solid boundary on the flow. A smooting of the forcing function usually occurs in the continuous forcing procedure since the immersed surface does not coincide with the grid points, so that this technique provides a rather diffusive representation of the immersed boundary.

On the contrary, in IB methods based on the discrete forcing technique, the effect of the boundary conditions on the immersed surface is taken into account without introducing a forcing term and its associated smoothing transfer, which allows for a sharp representation of the immersed boundary. For that, the computational stencil closed to the immersed boundary is usually modified to take into account the boundary conditions. We can mention three methods that fit in this category : -the ghost-cell technique based on the use of cells in the solid having at least one neighbor in the fluid (called ghost-cells) and on an interpolation scheme in order to enforce the boundary condition on the immersed surface -the cut-cell procedure in which the cells at the immersed boundary are truncated to create body conforming cells -and the hybrid Caresian/IB method in which the boundary conditions are enforced at the interior fluid nodes closest to the solid wall.

2.1 Continuous forcing IB methods

As previously written, elastic and rigid boundaries require different forcing terms, so that we present both cases in this section.

2.1.1 Elastic boundaries

This case corresponds to the original work on IB method performed by Peskin in the early 1970s [1]. In this work, Peskin tackled the coupled simulation of blood flow and muscle contraction of a heart considered as an elastic body. The immersed surface is represented by Lagrangian points of coordinate $\mathbf{X}_k$ moving with the fluid velocity and governed by the equation $\frac{\partial \mathbf{X}_k}{\partial t} = \mathbf{u}(\mathbf{X}_k, t)$.

The forcing term is based on the stress $\mathbf{F}$ of the elastic fibers representing the immersed boundary and on a smooth distribution function d which transfers the stress from Lagrangian points $\mathbf{X}_k$ to surrounding fluid nodes $\mathbf{x}_{ij}$ since they do not coincide with the immersed surface:

$$\mathbf{f}_m(\mathbf{x}_{ij}, t) = \sum_k \mathbf{F}_k(t) d(|\mathbf{x}_{ij} - \mathbf{X}_k|)$$

Various distribution functions d were proposed in past works [1, 2, 3, 4]. Some examples of such functions are given in Fig. 1.

2.1.2 Rigid boundaries

To take into account this category of boundaries, the structure is usually considered to be attached to an equilibrium location by a spring of restoring force $\mathbf{F}$ [2, 3]:

$$\mathbf{F}_k(t) = -\kappa(\mathbf{X}_k - \mathbf{X}_k^e(t))$$

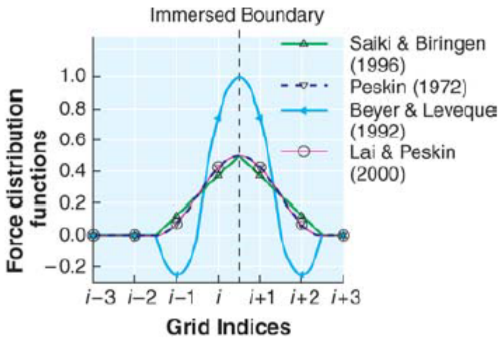

Figure 1: Distribution functions used in past works [36].

where $\mathbf{X}_k^e$ is the equilibrium location of the Lagrangien point $\mathbf{X}_k$, and κ is the spring constant whose positive value is large in order to accurately impose the boundary condition on the rigid immersed surface. It should be noted, however, that large values of κ result in a stiff system of equations subject to severe stability constraints [3], while lower values can lead to an excessive deviation from the equilibrium location [3].
One can mention the work of Goldstein et al. [5] in which the forcing term is given by

$$\mathbf{F}_k(t) = \alpha \int_0^t \mathbf{u}(\tau) d\tau + \beta \mathbf{u}(t)$$

where the parameters α and β are chosen to better enforce the boundary condition at the immersed surface. It should be noted again that large values of these parameters are required for accuratly enforcing the boundary conditions, which can lead to stability problems.
Another approach in this category is the penalization method in which the flow is assumed to occur in a porous medium of permeabiblity taking large values in the fluid and small values in the solid [9, 10], the flow being therefore governed by the Navier-Stokes/Brinkman equations [15]. In this method, an extra force term $\mathbf{F} = \frac{\mu}{K}\mathbf{u}$ is introduced in the Navier-Stokes equations, where K is the medium permeability. As K is small only in the solid regions, the force term is only active in the solid, ensuring zero value to the velocity $\mathbf{u}$.
One can also mention the Characteristic Based Volume Penalization Method [55] which extends the Brinkman penalization to generalized Neumann and Robin boundary conditions for hyperbolic and parabolic equations by introducing hyperbolic penalization terms with characteristics directing inward on solid obstacles.
Many developments and numerical expriments have been carried out using a penalty-based IB approach [51, 50, 53, 55, 52, 39, 40, 41, 42, 43, 44, 45, 46, 47, 57, 58, 59, 68, 69, 72].

2.1.3 Advantages/weak points

The continuous forcing approach is independant of the discretization scheme, simple to implement, can manage moving boundaries involving large motions, and numerous applications of these methods were performed in past works, for flows with immersed elastic boundaries [2, 7, 6, 8] and immersed rigid ones [3, 2, 5]. Nevertheless, and as noticed above, the smoothing of the forcing function does not allow a sharp representation of the immersed boundaries. On the other hand, this approach is often parameter dependant, and can be subject to stiffness/stability problems in the case of rigid boundaries. Furthermore, these methods require the solution of the governing equations inside the solid which can lead to a significant computation overload, particularly for high Reynolds number flows.

2.2 Discrete forcing IB methods

In this subsection, we focus on sharp-interface IB methods, including the ghost-cell approach, the hybrid Cartesian/IB technique and the cut-cell procedure. Unlike the continuous forcing approach, with these methods no forcing term is added in the governing equations and no smooting transfer is introduced avoiding the spread of the effect of the immersed boundary.

2.2.1 Ghost-cell approach

In this approach, the boundary condition is implicitly applied on the immersed surface through the use of ghost-cells which are solid cells having at least one neighbor in the fluid (see Fig. 2). On the other hand, the value of a given flow variable ψ at the ghost-cell nodes G

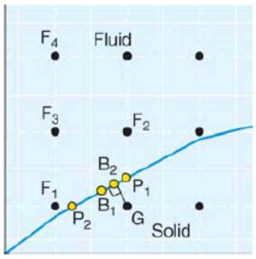

Figure 2: Ghost point (G), fluid points F_i, and boundary points B_i and P_i [36].

relies on an interpolation scheme which implicitly takes into account the boundary condition. For each ghost-cell G, the coefficients of the associated interpolation scheme are evaluated with the values of ψ at some fluid nodes F_i in the stencil and at some boundary nodes B_i and/or P_i (see Fig. 2, P_i are the intersection points between the x- or y-lines passing through G and the immersed surface, B_1 is the midpoint of P_1 and P_2, and B_2 is the intersection point between the normal passing through G and the immersed boundary), the number and the choice of these nodes depending on the interpolation order used.
This evaluation procedure in the ghost-cells is then performed simultaneously with the discretized governing equations for the fluid nodes, allowing to close these discrete equations.
Many developments and applications have been successfully carried out using this approach [11, 12, 13, 14, 16, 17, 18, 19, 20, 21, 22, 61, 66, 67, 70, 74, 75, 76].

2.2.2 Hybrid Cartesian/IB approach

Unlike the previous approach in which the forcing points are located in the solid domain (ghost-cells), in this sharp-interface IB approach, the boundary conditions are applied at the interior fluid nodes closest to the solid wall. The values of the flow variables at such nodes, noted B in this paragraph, are obtained by linear interpolation along an appropriate grid line between he nearest interior node to B and the point where the grid line intersects the solid wall where physical boundary conditions are known, as shown in Fig. 3 [23, 24]. A general interpolation scheme along the direction normal to the solid wall was proposed in [25]. This IB methdology, initially developped in the finite difference framework and for

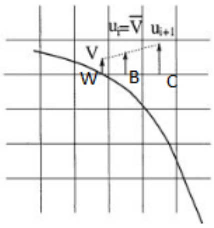

Figure 3: Linear interpolation of the velocity at the nearest-to-wall fluid node B using the velocity values at the nearest interior node to B, denoted C in the figure, and the intersection point W between the solid wall and the grid line carried by B and C [24].

incompressible flows, was extended for inviscid and viscous compressible flows in the finite volume framework in [26, 56]. The values of the flow variables at the interior fluid nodes B closest to the wall, where the boundary conditions are enforced, are obtained by linear interpolation in the wall-normal direction to close the discretized governing equations (Fig. 7).

2.2.3 Cut-cell approach

On the contrary to the above methods, the cut-cell methodology (also named Cartesian grid method) allows to satisfy the conservations laws for the cells which intersect the immersed surface. In this approach, the cells that cut the immersed boundary are truncated by keeping only the part that lies in the fluid. Among these cells, those whose center lies in the solid are gathered with neighboring cells (see Fig. 4a). The flux integrals on the faces of the reshaped cells are based on interpolated values of the flow variables. The coefficients of the interpolation function are evaluated using the values of the flow variables at appropriate points located in the fluid and at the immersed boundary (for example in Fig. 4b, the approximation of the flux f_{sw} is based on an interpolation using a six-point stencil).
Many developments and applications using the cut-cell technique have been carried out in past works [27, 38, 29, 30, 31, 16, 32, 33, 34, 35, 60, 62, 63, 64, 65, 71, 73].
It should be noted that the derivation of this approach to three dimensions is not trivial because the cut-cell technique leads to complex polyhedral cells on which the discretization of the Navier-Stokes equations is difficult.

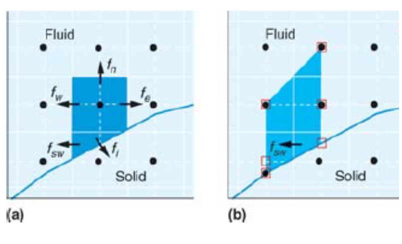

Figure 4: Cut-cell technique: (a) A reshaped cell at the immersed boundary. (b) Interpolation stencil for the evaluation of the flux f_{sw} [36].

2.2.4 Advantages/weak points

Unlike the continuous forcing approach, this category of IB methods allows for a sharp representation of the immersed boundaries, which makes it more suitable for relative high Reynolds number viscous flows (more accurate capture of boundary layer), and no extra stability constraints are encountered in the case of rigid bodies. Furthermore, discrete forcing IB techniques do not require the solution of the governing equations for the grid nodes located in the solid. One should also mention that, in its cut-cell version, the discrete forcing approach allows to preserve the conservation properties of the numerical scheme. Nevertheless, these methods introduce the boundary condition directly into the discrete equations and, from a practical point of view, their implementation is more complex than for the continuous forcing approach. On the other hand, the cut-cell approach can generate complex polyhedral cells in three dimensions, which makes the evaluation of the fluxes at the boundaries of the cells more difficult.
It should also be mentioned that taking into account moving boundaries with the discrete forcing technique is not as easy as for the continuous forcing term approach. Indeed, the flow variables in the cells located in the fluid which were in the solid at the previous time step due to the boundary motion, do not have a proper time history. One approach to solve this problem is to agglomerate these cells with neighboring fluid cells for the first time step after a solid cell becomes a fluid cell [37]. Another approach consists in evaluating the flow variables in these cells for one time step with an interpolation procedure based on neighboring fluid nodes and immersed boundary nodes [38].

3 Some recent works on the simulation of viscous compressible flows using IB methods

In this section, we focus on IB techniques developped for the simulation of compressible viscous flows since we are more particularly interested in these flows in the ANR project NORMA. The following works are presented according to increasing publication dates.

3.1 Work of Liu et al. [51]

Liu et al. propose a Brinkman penalization method for the computation of viscous compresible flows around solid obstacles. On the surface of the obstacle, the velocity satisfies the no-slip condition $\mathbf{u} = \mathbf{u}_s$ and the temperature is assumed constant $T = T_s$ where $\mathbf{u}_s$ and T_s are respectively, the velocity and the temperature of the obstacle. In their IB approach, the penalized terms involving the velocity and temperature are added to the momentum and energy equations. Furthermore, the continuity equation for porous media is considered inside obstacles in order to avoid nonphysical wave transmissions into obstacles. The penalized compressible Navier-Stokes equations in non-dimensional form are then written:

$$\left\{\begin{array}{rcl} \dfrac{\partial \rho}{\partial t} & = & -\left(1+(\dfrac{1}{\phi}-1)\chi\right)\nabla\cdot(\rho\mathbf{u}) \\ \dfrac{\partial \rho\mathbf{u}}{\partial t} & = & -\nabla\cdot(\rho\mathbf{u}\otimes\mathbf{u}) - \nabla p + \dfrac{1}{Re_a}\nabla\cdot\tau - \dfrac{\chi}{\eta}(\mathbf{u}-\mathbf{u}_s) \\ \dfrac{\partial E}{\partial t} & = & -\nabla\cdot[(E+p)\mathbf{u}] + \dfrac{1}{Re_a}\nabla\cdot(\mathbf{u}\tau) + \dfrac{1}{Re_a Pr(\gamma-1)}\nabla\cdot Q - \dfrac{\chi}{\eta_T}\rho(T-T_s) \end{array}\right.$$

where χ denote the characteristic function of the obstacle, ϕ is the porosity, $\eta = \alpha\phi$ is a normalized viscous permeability, $\eta_T = \alpha_T\phi$ is a normalized thermal permeability ($0 < \phi, \eta, \eta_T \ll$

1), Re_a is the acoustic Reynolds number, Pr being the Prandtl number, ρ, $\mathbf{u}$ and E denote the density, velocity and total energy, p is the pressure, $\tau = \mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T) - \frac{2}{3}\mu \nabla \cdot \mathbf{u} Id$, and $Q = \mu \nabla T$.
Amplitude and phase errors are estimated by asymptotic analysis, and show that the accuracy of the numerical solution can be controlled through the porosity ϕ.
The numerical applications concern a one-dimensional normal wave problem and a two dimensional cylindrical acoustic benchmark. The obtained results show a very good agreement with analytical solutions.

3.2 Work of Boiron et al. [50]

Boiron et al. proposed an IB method based on the Brinkman penalization technique for the simulation of viscous compressible flows around obstacles. In their work, they consider the two dimensional compressible Navier-Stokes equations with no-slip condition $\mathbf{u} = \mathbf{0}$ and a fixed temperature T_s on the surface of the obstacles. In the proposed IB model, these boundary conditions are enforced by adding penalization terms that involve directly the momentum and global energy unlike the model of Liu et al. [51], leading to the following modified Navier-Stokes equations :

$$\frac{\partial \mathbf{W}}{\partial t} + \nabla \cdot \mathbf{F}(\mathbf{W}) = \frac{1}{Re} \nabla \cdot \mathbf{F}_v(\mathbf{W}) - \frac{\chi}{\eta} \begin{pmatrix} 0 \\ \rho u \\ \rho v \\ E - \rho C_v T_s \end{pmatrix}$$

where χ is the characteristic function of the solid domain Ω_s and η the penalization parameter, $\mathbf{W} = (\rho, \rho u, \rho v, E)^t$ is the vector of conservative variables, ρ being the density, u and v the components of the velocity, E the total velocity, $\mathbf{F}$ and $\mathbf{F}_v$ representing the classical convective and viscous fluxes, and Re being the Reynolds number.
A formal asymptotic analysis according to ν is performed which shows that, by identifying the order η^0 terms, the initial Navier-Stokes equations is retrieved in the fluid domain, while the terms of order η^{-1} leads to the boundary conditions on the obstacles.
The spatial discretization is based on a high-resolution shock-capturing technique for the convective fluxes and on a fourth-order accurate centered finte difference scheme for the viscous fluxes. As for the time discretization, a third order TVD Runge-Kutta scheme is used. The numerical applications, which concern the IB computations on cartesian grids of two dimensional steady-state supersonic flows around a triangle and around one or several cylinders, show that the proposed penalization method can be used efficiently for the simulation of shocked flows with obstacles.

3.3 Work of Hu et al. [20]

In this work, a ghost cell technique is applied to simulate high Reynolds number compressible viscous flows on adaptative Cartesian grids. The two-dimensional compressible Navier-Stokes equations equipped with the SST-$k-\omega$ turbulence model is employed. The numerical model is a cell-centered, second-order accurate finite volume method. For the turbulent boundary condition, a wall function model is used. Hu et al. classified the cells into different categories based on their location relative to the immersed boundary (see Fig. 5). Then, the primitive flow variables at each ghost cell A (Fig. 5) are evaluated using a wall function and from the value of the flow variables at the associated reference point C (located on the normal to the immersed boundary passing through the ghost point and at a predetermined distance from this boundary, Fig. 5), as well as from different assumptions: the shear stress and the pressure are constant in the lower part of the boundary layer, non-permeable wall boundary condition, and the Crocco-Buseman equation. It should be noted that the value of the primitive variable at a reference point C is obtained by bilinear interpolation from the surrounding flow cells. As for the values of the turbulent variables k and ω at a ghost cell A, they are derived from the turbulent eddy viscosity at the associated reference point C and the friction velocity, both computed with a wall function model. On the other hand, the turbulent variables at the boundary cells (see Fig. 5) are also computed from their reference points using the same approach as for the ghost cells, while for the near-wall cells (see Fig. 5) the turbulent variables are evaluated by using a wall law model and the flow variables at these cells.

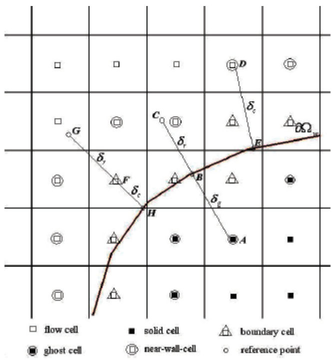

Figure 5: Flow cells, boundary cells, solid cells, ghost cells, near-wall cells, and reference points, as defined by Hu et al. [20].

The proposed methodology is applied to the simulation of two-dimensional turbulent flows over the RAE2822 airfoil in transonic regime and past a circular cylinder in the supersonic regime. It highlights the accuracy of the proposed IB method except in separated flow regions like those encountered in the cylinder flow problem.

3.4 Work of Abgrall et al. [53]

In this work, Abgrall et al. propose to combine the IB method with the level-sets technique and adaptated unstructured meshes. The IB method is based on the Brinkman penalization approach. The resulting modified compressible Navier-Stokes equations used in this work write:

$$\frac{\partial \mathbf{W}}{\partial t} + \nabla \cdot \mathbf{F}(\mathbf{W}) = \nabla \cdot \mathbf{F}_v(\mathbf{W}) + \frac{1}{\eta}\sum_{i=1}^{N_S} \chi_{S_i} \begin{pmatrix} 0 \\ \rho\mathbf{u} - \rho\mathbf{u}_{S_i} \\ \theta_{S_i}(\rho C_v T - \rho C_v T_{S_i}) + (\rho\mathbf{u} - \rho\mathbf{u}_{S_i}) \cdot \mathbf{u} \end{pmatrix}$$

where N_S is the number of rigid solids S_i, χ_{S_i} is the characteristic function of S_i and η the penalization parameter, $\mathbf{W} = (\rho, \rho\mathbf{u}, E)^t$ is the vector of conservative variables, ρ being the density, $\mathbf{u}$ the velocity, T the temperature, $\mathbf{F}$ and $\mathbf{F}_v$ representing the classical convective and viscous fluxes. At the boundary of a rigid solid S_i, the no-slip condition for the velocity $\mathbf{u} = \mathbf{u}_{S_i}$ is considered, and for the temperature T either the Dirichlet boundary condition $T = T_{S_i}$ (isothermal wall, $\theta_{S_i} = 1$ in the above equations) or Neumann boundary condition (adiabatic wall, $\theta_{S_i} = 0$ in the previous equations) can be applied.
In this above equations, χ_{S_i} is computed from a level set function ϕ_{S_i}: $\chi_{S_i} = H(-\phi_{S_i})$

where ϕ_{S_i} is the signed distance function to the boundary of S_i (ϕ_{S_i} is positive outside S_i and negative inside S_i) and H is the heavyside function. For moving bodies, ϕ_{S_i} satisfies the same advection equation as χ_{S_i}: $\frac{\partial \phi_{S_i}}{\partial t} + (\mathbf{u}_{S_i} \cdot \nabla)\phi_{S_i}$. In this work, an accurate description of the zero level set, i.e. the solid boundary, is obtained by an anisotropic mesh adaptation leading to a reduced mesh size near the solid boundaries and an improved accuracy of the IB method near the walls.
As for the numerical strategy, the spatial discretization is based on a classical mixed Finite Element/Finite Volume scheme using Roe's or HLLC numerical fluxes, and a Crank-Nicholson scheme is employed for the time discretization.
As for the numerical applications, a blasius test case (2D), a supersonic flow around a triangle (2D), a flow around a NACA0012 airfoil (2D) and a test case flow around an ellipse (3D) are performed. For a given accuracy, the proposed IB method is about 20% more expensive than its boddy-fitted counterpart but avoids the construction of meshes having to match complex geometries.

3.5 Work of Brown-Dymkoski et al. [55]

This paper presents an extension of Brinkman penalization methods to homogeneous/inhomogeneous Neumann and Robin boundary conditions for hyperbolic and parabolic equations. The proposed volume penalization technique, called Characteristic Based Volume Penalization Method, introduces hyperbolic penalization terms whose characteristics are directed inward along the normal direction to the body surface.
Hereinafter, it is shown how this penalization method applies to the non-dimensional compressible Navier-Stokes equations:

$$\left\{\begin{array}{rcl} \dfrac{\partial \rho}{\partial t} & = & -\nabla \cdot (\rho \mathbf{u}) \\ \dfrac{\partial \rho \mathbf{u}}{\partial t} & = & -\nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) - \nabla p + \dfrac{1}{Re_a} \nabla \cdot \tau \\ \dfrac{\partial E}{\partial t} & = & -\nabla \cdot [(E+p)\mathbf{u}] + \dfrac{1}{Re_a} \nabla \cdot (\mathbf{u}\tau) + \dfrac{1}{Re_a Pr(\gamma-1)} \nabla \cdot Q \end{array}\right.$$

where Re_a is the acoustic Reynolds number, Pr being the Prandtl number, ρ, $\mathbf{u}$ and E denote the density, velocity and total energy, p is the pressure, $\tau = \mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T) - \frac{2}{3}\mu \nabla \cdot \mathbf{u} Id$, and $Q = \mu \nabla T$.
Let RHS be the right hand side of the previous set of equations. As an example, it is assumed in the following that the no-slip condition $\mathbf{u} = \mathbf{u}_o$ and the heat flux condition $\frac{\partial T}{\partial \mathbf{n}} = q$ are applied on the surface of an obstacle O, where T is the temperature, $\mathbf{n}$ denotes the inward-oriented surface normal, $\mathbf{u}_o = \mathbf{u}_o(\mathbf{x}, t)$ and $q = q(\mathbf{x}, t)$ being respectively the velocity of the obstacle O and the heat flux to be imposed at the wall of O.
The penalized Navier-Stokes equations then write in a domain Ω containing the obstacle O:

$$\left\{\begin{array}{rcl} \dfrac{\partial \rho}{\partial t} & = & (1-\chi) \times RHS_\rho - \dfrac{\chi}{\eta_c}\left(\dfrac{\partial \rho}{\partial \mathbf{n}} - \phi\right) \\ \dfrac{\partial \rho \mathbf{u}}{\partial t} & = & (1-\chi) \times RHS_{\rho \mathbf{u}} - \chi\left(\dfrac{1}{\eta_b}\rho(\mathbf{u}-\mathbf{u}_o) - \rho \nu_n \dfrac{\partial^2 \mathbf{u}}{\partial x_j \partial x_j} + \dfrac{1}{\eta_c}\mathbf{u}\left(\dfrac{\partial \rho}{\partial \mathbf{n}} - \phi\right)\right) \\ \dfrac{\partial E}{\partial t} & = & (1-\chi) \times RHS_E - \chi\left(\dfrac{1}{\eta_c}\dfrac{\partial E}{\partial \mathbf{n}} + \dfrac{\rho(\mathbf{u}-\mathbf{u}_o)\cdot \mathbf{u}}{\eta_b} - \dfrac{\rho \mathbf{u}}{\eta_c} \cdot \dfrac{\partial \mathbf{u}}{\partial \mathbf{n}} - \rho \mathbf{u} \cdot \nu_n \dfrac{\partial^2 \mathbf{u}}{\partial x_j \partial x_j} \right. \\ & & \left. -\dfrac{1}{\eta_c}\dfrac{E}{\rho}\phi - \dfrac{1}{\eta_c} c_v \rho q\right) \end{array}\right.$$

where χ denotes the characteristic function of the object O, η_b and η_c are the penalization parameters (η_b and $\eta_c \to 0$, optimally $\eta_b < \eta_c$), ν_n is a non-physical diffusion which avoids the creation of a discontinuous solution across the obstacle boundary (for a resolution of Δx within O, the diffusive coefficient must be $\nu_n \geq O(\Delta x^2/\eta_b)$). Since the volume of the obstacle is penalized and not its surface, the inward-oriented surface normal $\mathbf{n}$ is defined everywhere by linear extension throughout O. In the above penalized equations, the quantity ϕ satisfies $\dfrac{\partial \phi}{\partial t} = -\dfrac{\chi}{\eta_c}\dfrac{\partial \phi}{\partial \mathbf{n}}$ where $\phi = (1-\chi)\dfrac{\partial \rho}{\partial \mathbf{n}} + \chi\phi$. Due to the inward pointing characteristic, the definition of ϕ in the whole domain Ω provides the necessary boundary condition for this hyperbolic equation on ϕ, which is solved only within O. In other words, the density derivatives are physically determined outside the obstacle O with the continuity equation and extrapolating inside O by integrating this additional equation on ϕ.
It should be noted that in the penalized region, ϕ becomes the target for the inhomogeneous Neuman condition on ρ, yielding to the penalized continuity equation given above. An appropriate penalized equation for ρ must be provided, otherwise the equations of state are under-constrained, and, on the other hand, ρ is solved in the whole domain Ω.
It should also be noted that, unlike the Brinkman penalization, this volume penalization approach does not retain the constitutive equations inside the obstacle, which also explains the presence of a non-physical diffusion term in the penalized equation (due to the Dirichlet condition on the velocity $\mathbf{u}$). It is also important to note that, with the definition of the normal $\mathbf{n}$, the hyperbolic penalization terms (for Neumann and Robin conditions) has inward-pointing charateristics that extend perpendicular to the surface into the obstacle O. The solution then propagates from the surface inward, which enforces the desired derivative. It also prevents a non-physical solution in O from spreading outward, and then the solution in the domain outside O is only affected through the surface derivative imposed by penalization.
In this work, the penalized Navier-Stokes equations are discretized using a numerical solver based on a wavelet decomposition to dynamically adapt on steep gradients in the solution while retaining a predetermined order of accuracy (Adaptive Wavelet Collocation Method). Adaptative mesh refinement grids are also used in order to have high resolution around surfaces for computational accuracy and a proper definition of the geometry, as volume penalization techniques do not require body-conformal meshes.
The proposed penalization approach was successfully applied to a one-dimensional diffusion problem, a one-dimensional acoustic reflection benchmark, flows past a two-dimensional circular cylinder at Reynolds numbers 40 (steady flow) and 1000 (unsteady flow with vortex shedding), and the flow past a moving two-dimensional circular cylinder at Reynolds number 185. As with Brinkman penalization methods, the error was shown to be rigorously controlled by the penalization parameters.

3.6 Work of Piquet et al. [52]

This paper compares the Brinkman penalization approach with the direct forcing IB method for the simulation of compressible viscous flows. On the surface of the obstacle, the no-slip condition $\mathbf{u} = \mathbf{u}_s$ applies for the fluid velocity and the temperature is assumed constant $T = T_s$ where $\mathbf{u}_s$ and T_s are respectively the velocity and the wall temperature of the obstacle.
As for the space and time discretization, a fifth-order WENO scheme and a third-order TVD Runge-Kutta scheme are used.

The penalized Navier-Stokes equations used in this work write:

$$\left\{\begin{array}{lcl} \dfrac{\partial \rho}{\partial t} & = & -\left(1+(\dfrac{1}{\phi}-1)\chi\right)\nabla\cdot(\rho\mathbf{u}) \\ \dfrac{\partial \rho\mathbf{u}}{\partial t} & = & -\nabla\cdot(\rho\mathbf{u}\otimes\mathbf{u})-\nabla p+\nabla\cdot\tau-\dfrac{\chi}{\eta}(\rho\mathbf{u}-(\rho\mathbf{u})_s) \\ \dfrac{\partial E}{\partial t} & = & -\nabla\cdot[(E+p)\mathbf{u}]+\nabla\cdot(\mathbf{u}\tau)+\dfrac{c_p}{Pr}\nabla\cdot Q-\dfrac{\chi}{\eta}(E-E_s) \end{array}\right.$$

where χ denote the characteristic function of the solid, ϕ is the porosity, η is the permeability ($0 < \phi, \eta \ll 1$), c_p is the heat capacity at constant pressure, Pr being the Prandtl number, ρ, $\mathbf{u}$ and E denote the density, velocity and total energy, $\tau = \mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T) - \frac{2}{3}\mu\nabla\cdot\mathbf{u}Id$, and $Q = \mu\nabla T$.

The direct forcing approach proposed in this work uses a Cartesian grid split into fluid, solid and ghost cells as defined in Fig. 6a. For each ghost point G, an image point IP is created in the fluid domain as depicted in Fig. 6b. The bi-linear interpolation method is used to interpolate the value at the image point IP from the neighboring fluid points and possibly some points located on the immersed boundary if IP is surrounded by one or more ghost points (6b). In order to evaluate the weighting coefficients C_i of the bi-linear interpolation method given for a generic flow variable ϕ by $\phi(x, y) = C_1xy + C_2x + C_3y + C_4$ in 2D and $\phi(x, y, z) = C_1xyz + C_2xy + C_3xz + C_4yz + C_5x + C_6y + C_7z + C_8$ in 3D, a 4×4 (2D) or 8×8 (3D) Vandermonde matrix is inverted for each ghost point. Once the interpolated values ϕ_{IP} at the image point IP of the flow variables ϕ are computed, the boundary conditions at the immersed surface $\mathbf{u} = \mathbf{u}_s$, $T = T_s$ and $\frac{\partial p}{\partial n} = 0$ lead to the following values of $\mathbf{u}_G$, T_G and p_G at the ghost point G: $\mathbf{u}_G = 2\mathbf{u}_s - \mathbf{u}_{IP}$, $T_G = 2T_s - T_{IP}$ and $p_G = p_{IP}$.

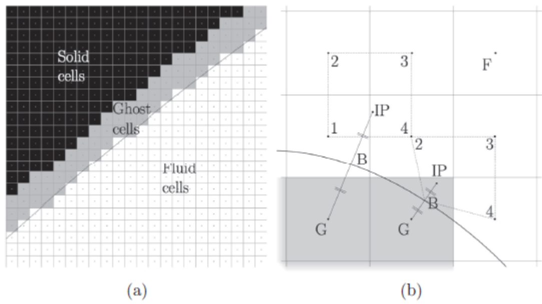

Figure 6: (a) Fluid cells, solid cells, ghost cells, as defined by Piquet et al. (b) Bi-linear interpolation variables at image points IP from the neighboring fluid nodes and possibly interface points B [52].

As numerical applications, a one-dimensional normal shock reflection off a solid wall, a two dimensional transonic shock/boundary layer in a viscous shock tube, a two-dimensional supersonic shock/cylinder interaction, and a three-dimensional supersonic turbulent channel flow are considered. The comparison shows that, with sufficient mesh resolution, the Brinkman penalization approach and the ghost-cell method yield qualitatively similar results. The penalization technique is found to be an accurate and efficient method, though it suffers from a lack of regularity in the very near-wall pressure fluctuations, which is attributed to the fact that no specific pressure condition at the fluid/solid interface is required. It was also found that the ghost-cell approach is much more complex to implement compared to the penalization method, and the Vandermonde matrix has to be recomputed at every time step in the case of moving objects, which makes this discrete forcing approach less efficient in terms of computational cost.

3.7 Work of Pu et al. [56]

In this work, a discrete forcing IB method is developped for the simulation of compressible turbulent flows using RANS models and wall modeling. In the proposed IB approach, the flow variables at fluid nodes located in the immediate vicinity of a wall (called IB nodes, Fig. 7) are evaluated by linear interpolation in the direction normal to the wall in order to close the discretized governing equations. For this purpose, fictitious points (may not be mesh nodes) inside the fluid domain are introduced (points F,F_1,F_2 in Fig. 7). All flow variables at a fictitious point are computed by linear interpolation over the triangle containing it. If W denotes the normal-wall intersection for a given IB node B (see Fig.7), then the normal velocity at B is obtained by linear interpolation between points W and F and by applying the no-penetration boundary condition. The pressure at the IB node B is obtained by considering a simpified inviscid momentum equation in the direction normal to the wall and by using the value of the pressure and density at the corresponding fictitious point F. The tangential velocity at B is evaluated via the wall shear stress prescribed by the wall modeling technique and by using the flow variables at point F. On the other hand, the density at the IB node B is deduced from the temperature at B given by the Crocco-Busemann relation using the flow variables at F. As for the turbulent variables, and considering for example the SST $k-\omega$ model, ω is fisrt evaluated at point F by interpolation between a viscous sublayer value and a log-layer value of ω, and then the value of ω at node B is set equal to that at point F. At least, k is obtained at the IB node B by using the value of ω at B and the profile of the eddy viscosity in the vicinity of the wall.

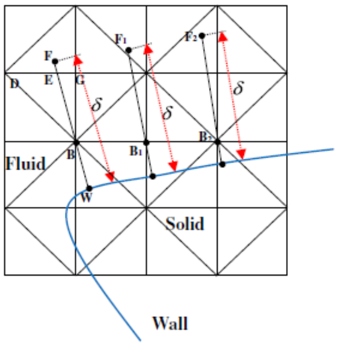

Figure 7: IB model of Pu et al. [56]. B,B_1,B_2: IB nodes; F,F_1,F_2: fictitious points; for an IB node B, F is located on the normal to the wall passing through B and located at a distance δ from the wall (computed as the maximal distance between W and E over all IB nodes B).

The governing equations are discretized by the Galerkin finite element approach and the resulting semi-discrete equations are integrated in time via a dual-time stepping scheme.

As for the numerical experiments, the test cases investigated in this work are a flat plate, the NACA0012 airfoil (fixed and oscillating), the RAE2822 airfoil, and the Onera M6 wing (3D). The RANS SST $k-\omega$ or Spalart-Almaras model is employed in the simulations. The used triangular and tetrahedral meshes are derived from Cartesian meshes (by dividing rectangular or tetrahedral cells, respectively), and are refined locally in the region near the solid boundary. The proposed IB/wall modeling approach yields acceptable results for the prediction of high-Reynolds compressible turbulent flows past fixed and moving bodies.

3.8 Work of Abalakin et al. [39]

Alabakin et al. developped a continuous forcing approach based on the Brinkman penalization technique for the simulation of a three dimensional flow around a circular cylinder using the IDDES hybrid turbulence approach derived from the Spalart-Allmaras model. If χ is the characteristic function of the solid domain ($\chi = 1$ in the solid, boundary included, and 0 elsewhere) and η the small penalization parameter, the Navier-Stokes equations equipped with the Spalart-Allmaras equation in the IDDES framework and modified by the penalization terms writes in this work:

$$\left\{\begin{array}{rcl} \dfrac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) & = & 0, \\ \dfrac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) & = & -\nabla p + \mu \nabla \cdot \tau + \nabla \cdot (\rho \nu f_{\nu_1} \tau) - \dfrac{\chi}{\eta} \rho \mathbf{u}, \\ \dfrac{\partial E}{\partial t} + \nabla \cdot [(E+p)\mathbf{u}] & = & \mu \nabla \cdot (\mathbf{u}\tau) + \nabla \cdot (\rho \nu f_{\nu_1} \mathbf{u}\tau) + \nabla \cdot Q - \dfrac{\chi}{\eta} \rho ||\mathbf{u}||^2 \\ \dfrac{\partial \rho \nu}{\partial t} + \nabla \cdot (\rho \nu \mathbf{u}) & = & D_\nu + G_\nu - c_\omega f_\omega \left(\dfrac{\nu}{l_T}\right)^2 - \dfrac{\chi}{\eta} \rho \nu \end{array}\right.$$

where ρ, $\mathbf{u}$ and E denote the density, velocity and total energy, μ is the molecular viscosity, $\tau = (\nabla \mathbf{u} + \nabla \mathbf{u}^T) - \frac{2}{3} \nabla \cdot \mathbf{u} Id$, $Q = C_v \left(\mu + \rho \nu f_{\nu_1} + \frac{\gamma}{P_r} \mu\right) \nabla T$, and in the last equation governing the turbulent kinematic eddy viscosity ν:

- D_ν and G_ν are respectively the diffusion and production of turbulence (given, for instance, in [48]),
- f_ω, f_{ν_1} denote damping functions (can be found, for instance, in [48])
- l_T is the turbulence scale introcuced by the IDDES model (described, for instance, in [49]).

The penalization source terms in the above system of equations ensure a fast relaxation of order $\exp(-t/\eta)$ of the solution for the velocity $\mathbf{u}$ and the turbulent kinematic eddy viscosity ν to the zero Dirichlet condition at the grid nodes for which the characteristic function is equal to one. An edge-based reconstruction finite volume scheme and an implicit second-order time integration scheme are used for the discretization of the above penalized system on unstructured grids.
The turbulent flow past a three-dimensional circular cylinder at Reynolds number 3900 is performed using the proposed IB method on unstructured grids. The obtained results show that the IB techniques is in good agreement with the traditional approach based on body fitted grid.

4 Conclusion

A review on IB methods is proposed in this document. Some important works have been reviewed, ranking from that of Peskin [1] in 1972 on the simulation of blood flow and heartbeat using the IB technique to recent works on IB methods with applications in turbulence [20, 56, 39]. The main IB techniques are briefly presented, including penalty-based methods, the ghost-cell procedure, the cut-cell technique and the hybrid Cartesian/IB approach. The strengths and weaknesses of the different methods are recalled. On the basis of this review, it can be said that few works on IB methods were conducted with three-dimensional compressible turbulent flows, and particularly in the case of moving geometries. In the field of CFD, there is therefore a need to develop numerical methods which allow for a good prediction of three-dimensional turbulent flows on grids non conforming to the body, with possibly moving geometries, including in particular the accurate capture of boundary layers which remains a challenging problem.

References

[1] C.S. Peskin, Flow patterns around heart valves: A numerical method, *Journal of Computational Physics*, 10(2):252-271, 1972.

[2] R.P. Beyer, R.J. Leveque, Analysis of a one-dimesional model for the immersed boundary method, SIAM J. Numer. Anal. 29:332-364, 1992.

[3] M.C. Lai, C.S. Peskin, An immersed boundary method with formal second-order accuracy and reduced numarical viscosity, J. Comput. Phys., 160:705-719, 2000.

[4] E.M. Saiki, S. Biringen, Numerical simulation of a cylinder in uniform flow: applicationof a virtual boundary method, J. Comput. Phys., 123:450-465, 1996.

[5] D. Goldstein, R. Handler, L. Sirovich, Modeling a no-slip flow boundary boundary with an external force field, J. Comput. Phys., 105:354-366, 1993.

[6] C.S. Peskin, The fluid dynamics of heart valves: experimental, theoritical and compitational methods, Annu. Rev. Fluid Mech., 14:235-259, 1981.

[7] L.J. Fauci, A. McDonald, Sperm mobility in the presence of boundaries, Bull. Math. Biol., 57:679-699, 1994.

[8] S. Unverdi, G. Tryggvason, A front-tracking method for viscous incompressible, multifluid flows, J. Comput. Phys., 100:25-42, 1992.

[9] P. Angot, C.H. Bruneau, P. Fabrie, A penalization method to take into account obstacles in viscous flows, Numer. Math., 81:497-520, 1999.

[10] K. Khadra, P. Angot, S. Parneix, J.P. Caltagirone, Fictitious domain approach for numerical modeling of Navier-Stokes equations, Int. J. Numer. Methods Fluids, 34:651-684, 2000.

[11] S. Majumbar, G. Iaccarino, P. Durbin, RANS solvers with adaptive structured boundary non-conforming grids, Center for Turbulence Research Briefs, pages 353-366, 2001.

[12] Y.H. Tseng, J.H. Ferziger, A ghost-cell immersed boundary method for flow in complex geometry, J. Comput. Phys., 192(2):593-623, 2003.

[13] A. Dadone, B. Grossman, Ghost-cell method for analysis of inviscid three-dimensional flows on Cartesian-grids, Comput. Fluids, 36:1513-1528.

[14] R. Ghias, R. Mittal, T.S. Lund, A non-body conformal grid method for simulation of compressible flows with complex immersed boundaries, AIAA Pap., 2004-0080.

[15] H.C. Brinkman, A calculation of the viscous force exerted by a flowing fluid on a swarm of particles, Appl. Sci. Res. 1:27-34, 1947.

[16] R. Mittal, V. Seshadri, H.S. Udaykumar, Flutter, tumble and vortex induced autorotation, Theor. Comput. Fluid. Dyn., 17(3):165-170, 2004.

[17] R. Mittal, H. Dong, M. Borzkurttas, F.M. Najjar, A. Vargas, A. von Loebbecke, A versatile sharp interface immersed boundary method for incompressible flows with complex geometries, J. Comput. Phys., 227(10):4825-4852, 2008.

[18] G. Iaccarino, G. Kalitzin, C.J. Elkins, Numerical and experimental investigation of the turbulent flow in ribbed serpentine passage, Annu. Res. Briefs, Cent. Turbul. Res., pp. 379-388, 2003.

[19] G. Kalitzin, G. Iaccarino, B. Khalighi, Towards an immersed boundary solver for RANS simulation, AIAA pap., 2003-0770.

[20] O. Hu, N. Zhao, J.M. Liu, A ghost cell method for turbulent compressible viscous flows on adaptative Cartesian grids, 7th Asian-Pacific Conference on Aerospace Technology and Science, 7th APCATS 2013.

[21] D. Pan, T.T. Shen, Computation of incompressible flows with immersed bodies by a simple ghost cell method, Int. J. Numer. Meth. Fluids, 60(12):1378-1401, 2008.

[22] A.F. Shinn, M.A. Goodwin, S.P. Vanka, Immersed boundary computation of shear- and buoyancy-driven flows in complex enclosures, International Journal of Heat and Mass Transfer, (52)(17-18):4082-4089, 2009.

[23] J. Mohd-Yusof, Combined immersed boundaries/B-splines methods for simulations of flows in complex geometries, CTR Annual Research Briefs, 317-327, 1997.

[24] E.A. Fadlun, R.Verzicco, P. Orlandi, J. Mohd-Yusof, Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations, J. Comput. Phys., 161:35-60, 2000.

[25] A. Gilmanov, F. Sotiropoulos, E. Balaras, A general reconstruction algorithm for simulating flows with complex 3D immersed boundaries on Cartesian grids, J. Comput. Phys., 191:660-669, 2003.

[26] Y. Zhang, C.H. Zhou, An immersed boundary method for simulation of inviscid compressible flows, Int. J. Numer. Methods Fluids, 74:775:793, 2014.

[27] T. Ye, R. Mittal, H.S. Udaykumar, W. Shyy, An accurate Cartesian Grid method for Viscous Incompressible Flows with Complex Immersed Boundaries, J. Comput. Phys., 156(2):209-240, 1999.

[28] H.S. Udaykumar, R. Mittal, P. Rampungoon, A. Khanna, A Sharp Interface Cartesian Grid Method for Simulating Flows with Complex Moving Boundaries, J. Comput. Phys., 147(1):345-380, 2001.

[29] H.S. Udaykumar, R. Mittal, P. Rampungoon, Interface tracking finite volume method for complex solid-fluid intercations on fixed meshes, Commun. Numer. Meth. Engng., 18(2):89-97, 2002.

[30] R. Mittal, C. Bonilla, H.S. Udaykumar, Cartesian grid methods for simulating flows with moving boundaries, In computational Methods and Experimental Measurements-XI, ed. C.A. Brebbia, G.M. Carlomagno, P. Anagnostopoulos, 2003.

[31] R. Mittal, Y. Utturkar, H.S. Udaykumar, Computational modeling and analysis of biomimetic flight mechanisms, AIAA Pap., 2002-0865.

[32] Y. Utturkar, R. Mittal, Sensitivity of synthetic jets to the design of the jet cavity, AIAA Pap. 2002-0124.

[33] M.P. Kirkpatrick, S.W. Armfield, J.H. Kent, A representation of curved boundaries for the solution of the Navier-Stokes equations on a staggered three-dimensional Cartesian grid, J. Comput. Phys., 184(1):1-36, 2003.

[34] M. Dröge, R. Verspatten, A new symmetry-preserving Cartesian-grid method for computing flow past arbitrarily shaped objects, Int. J. Numer. Meth. Fluids, 47(8-9):979-985, 2005.

[35] M.H. Chung, Cartesian cut cell approach for simulating incompressible flows with rigid bodies of arbitrary shape, Computer & Fluids, 35(6):607-623, 2006.

[36] R. Mittal, G. Iaccarino, Immersed Boundary Methods,Ann. Rev. Fluid. Mech., 37:239-261, 2005.

[37] H.S. Udaykumar, R. Mittal, W. Shyy, Computation of solid-liquid phase fronts in the sharp interface limit on fixed grids, J. Comput. Phys., 153:534-574, 1999.

[38] H.S. Udaykumar, R. Mittal, P. Rampunggoon, A. Khanna, A sharp interface Cartesian grid method for simulating flows with complex moving boundaries, J. Comput. Phys., 174:345-380, 2001.

[39] I.V. Abalakin, A.P. Duben, N. S. Zhdanova, and T.K. Kozubskaya, Simulating an Unsteady Turbulent Flow around a Cylinder by the immersed Boundary Method, Mathematical Models and Computer Simulations, 2019, Vol.11, No.1, pp. 74-85.

[40] I.V. Abalakin, N.S. Zhdanova, T.K. Kozubskaya, Immersed boundary method implemented for the simulation of an external flow on unstructured meshes, Math. Mod. and Comp. Simul., 8(3), pp. 219-230, 2016.

[41] A.P. Duben, N.S. Zhdanova, T.K. Kozubskaya, Numerical investigation of the deflector effect on the aerodynamic and acoustic characteristics of turbulent cavity flow, Fluid Dyn., 2017, 52(4), pp. 561-571, 2017.

[42] N.S. Zhdanova, A.V. Gorobets, V. Abalakin, Supercomputer simulations of fluid-structure interaction problems using an immersed boundary method, Supercomputing Frontiers and Innovations (SuperFrI), 5(4), pp. 78-82, 2018.

[43] I.V. Abalakin, N.S. Zhdanova, T.K. Kozubskaya, Immersed Boundary Method for Numerical Simulation of Inviscid Compressible Flows, Comp. Math. Math. Phys., 58(9), pp. 1411-1419, 2018.

[44] I.V. Abalakin, A.P. Duben, T.K. Kozubskaya, L.N. Kudryavtseva, N.S. Zhdanova, Immersed Boundary Method on Deformable Unstructured Meshes for Airfoil Aeroacoustic Simulation, Computational Mathematics and Mathematical Physics, 59(12), pp. 1982-1993, 2019.

[45] I.V. Abalakin, V.A. Vershkov, N.S. Zhdanova, T.K. Kozubskaya, Numerical simulation of acoustic fields induced by flow past oscillating solid, Math. Mod. and Comp. Simul., V. 12, (3), pp. 422-432, 2020.

[46] V.O. Tsvetkova, T.K. Kozubskaya, L.N. Kudryavtseva, N.S. Zhdanova, On mesh adaptation for supercomputer simulation of flows around solid bodies defined by immersed boundary method, Procedia Computer Science, V. 178, pp. 404-413, 2020.

[47] I.V. Abalakin, N.S. Zhdanova, T.K. Kozubskaya, Immersed Boundary Penalty Method âĂŞ In Proceedings of 6th European Conference on Computational Mechanics (ECCM 6) & 7th European Conference on Computational Fluid Dynamics (ECFD 7), Glasgow, UK, 11-15 June 2018.

[48] P.R. Spalart, S.R. Allmaras, A one-equation turbulence model for aerodynamic flows, Rech. Aerospat., No 1, 5-21, 1994.

[49] M.L. Shur, P.R. Spalart, M.K. Strelets, A.K. Travin, A hybrid RANS-LES approach with delayed-DES and wall modeled LES capabilities, Int. J. Heat Fluid Flow, 29:1638-1649, 2008.

[50] O. Boiron, G. Chiavassa, R. Donat, A high-resolution penalization method for large Mach number flows in the presence of obstacles, Computer & Fluids, 38:703-714, 2009.

[51] Q. Liu, V. Vasilyev, A Brinkman penalization method for compressible flows in complex geometries, J. Comput. Phys., 227:946-966, 2007.

[52] A. Piquet, O. Roussel, A. Hadjadj, A comprative study of Brinkman penalization and direct-forcing immersed boundary methods for compressible viscous flows, Comuter and Fluids, 136:272-284, 2016.

[53] R. Abgrall, H. Beaugendre, C. Dobrzynski, An immersed boundary method using unstructured anisotropic mesh adaptation combined with level-sets and penalization techniques, Journal of Computational Physics, 257:83-101, 2014.

[54]

[55] E. Brown-Dymkoski, N. Kasimov, O. V. Vasilyev, A characteristic based volume penalization method for general evolution problems applied to compressible viscous flows, Journal of Computational Physics, 262:344-357, 2014.

[56] T.M. Pu, C.H. Zhou, An immersed boundary/wall modeling method for RANS simulation of compressible turbulent flows, In. J. Numer. Math. Fluids, 87:215-238, 2018.

[57] E. Brown-Dymkoski, N. Kasimov, O.V. Vasilyev, A characteristic based volume penalization method for general evolution problems applied to compressible viscous flows, J. Comput. Phys., V. 262, pp. 344-357, 2014.

[58] E.Feireisl, J.Neustupa, J.Stebel, Convergence of a Brinkman-type penalisation fo compressible fluid flows, Journal of Differential Equations 250:596-606, 2011.

[59] I.V.Abalakin, N.S.Zhdanova, T.K.Kozubskaya, Immersed boundary method as applied simulation of external aerodynamics problems with various boundary conditions, Dokl. Math. 91, 178-181, 2015.

[60] P.Tucker, Z.Pan, A cartesian cut cell method for incompressible viscous flow, Applied Mathematical Modelling 24:591-606, 2000.

[61] J.W.Nam, F.S.Lien, A ghost-cell immersed boundary method for large eddy simulations of compressible turbulent flows, International Journal of Computationnal Fluid Dynamics, 2014.

[62] J.W.Nam, F.S.Lien, Assessment of ghost-cell based cut-cell method for large eddy simulations of compressible flows at hight Reynolds number, Internationnal Journal of Heat and Fluid Flow 53:1-14, 2015.

[63] Z.Xie, T.Stroesser, A three- dimensionnal cartesian cut-cell/volume of fluid method for two phase flows with moving bodies, Journal of Computationnal Physics, 416 :109536, 2020.

[64] B.Muralidharan, S.Menon, Simulation of moving boundaries interacting with compressible reacting flows using a second order adaptative cartesian cut-cell method, Journal of Computationnal Physics, 357:230-262, 2018.

[65] N.Gokale, N.Nikiforakis, R.Klein, A dimensionnaly split Cartesian cut cell method for the compressible Navier-Stokes equations, Jounal of Computationnal Physics 375:1205-1219, 2018.

[66] C.Yan, W.X.Huang, G.X.Lui, C.Xu, Z.S.Zhung, A ghost-cell immersed boundary method for large eddy simulation of flows in complex geometries, International Journal of Computationnal Fluid Dynamics, 2015.

[67] Y.Mizuno, S.Takahashi, T.Nonomura, T.Nagata, K.Fukuda, A simple immersed boundary method for compressible flow simulation arround a stationary and mouving sphere, Mathematical Problems in Engineering, 2015.

[68] A.Sarthou, S.Vincent, P.Angot, J.P.Caltagirone, The sub-mesh penalty method, 5thInternational Symposium on Finite Volumes for Complex Applications, 2008.

[69] W. Bizid, A. Etcheverlepo, S. Vincent, J.P. Caltagirone, D.Monfort, M. Hassine, Penalty methods for turbulent flows interacting with obstacles, M.O. Deville et al. (eds.), Turbulence and Interactions, Notes on Numerical Fluid Mechanics and Multidisciplinary Design 125, 2014.

[70] J.R. Aarnes, T. Jin, C. Mao, N.E.L. Haugen, K. Luo, H. I. Andersson, Treatment of solid objects in the Pencil Code using an immersed boundary method and overset grids, Geophysical & Astrophysical Fluid Dynamics, 114(1-2):1-23, 2018.

[71] N. James, E. Maitre, I. Mortazavi, Immersed boundary methods for the numerical simulation of incompressible aerodynamic and fluid-structure interactions, Annales Mathématiques Blaise Pascal, January 2013.

[72] K. Schneider, Immersed boundary methods for numerical simulation of confined fluid and plasma turbulence in complex geometries: a review, J. Plasma Phys., vol. 81, 435810601, 2015.

[73] Sinan Q. Salih, Mohammed Suleman Aldlemy, Mohammad Rasidi Rasani,A. K. Ariffin, Tuan Mohammad Yusoff Shah Tuan Ya, Nadhir Al-Ansari, Zaher Mundher Yaseen, Kwok-Wing Chau, Thin and sharp edges bodies-fluid interaction simulation using cutcellimmersed boundary method, Engineering Applications of Computational Fluid Mechanics, 13:1,860-877, 2019.

[74] J.H. Seo, R. Mittal, A High-Order Immersed Boundary Method for Acoustic Wave Scattering and Low-Mach Number Flow-Induced Sound in Complex Geometries, Journal of Computational Physics 230(4):1000-1019, 2011.

[75] M.E. Khalili, M. Larsson, B. Müller, Immersed boundary method for viscous compressible flows around moving bodies, Volume 170, Pages 77-92, 2018.

[76] R. Ghias, R. Mittal, and H. Dong, A sharp interface immersed boundary method for compressible viscous flows, Journal of Computational Physics, vol.225, issue.1, pp.528-553, 2007.