
Yoann Cheny & Olivier Botella

Nancy Universités
LEMTA - UMR 7563 (CNRS-INPL-UHP)
2, avenue de la Forêt de Haye, B.P. 160, 54504 Vandœuvre-lès-Nancy (France)
Yoann.Cheny@ensem.inpl-nancy.fr ; Olivier.Botella@ensem.inpl-nancy.fr

Abstract:

This paper concerns the development of a new Cartesian grid / immersed boundary (IB) method for the computation of incompressible viscous flows in irregular geometries. In IB methods, the irregular boundary is not aligned with the computational grid, and of utmost importance for accuracy and stability is the discretization in cells which are cut by the boundary, the so-called “cut-cells”. In this paper, we present a progress report on a new finite volume discretization where the irregular boundary is implicitly represented by its signed distance function (the level-set function). With the help of level-set calculus tools, we are able to preserve the Cartesian structure of the discretized Navier-Stokes equations in the cut-cells. The resulting discrete systems are efficiently solved with a black box multigrid solver for structured grids. The method is validated for the circular cylinder flow at low Reynolds number.

Key-words:

Incompressible flows ; Computational fluid dynamics ; Immersed boundary methods

1 Introduction

Much attention has recently been devoted to the extension of Cartesian grid flow solvers to complex geometries by immersed boundary (IB) methods (see [7] for a recent review). In these methods, the irregular boundary is not aligned with the computational grid, and the treatment of the cells which are cut by the boundary remains an important issue. Indeed, the discretization in these cut-cells should be designed such that: (a) the overall accuracy of the method is not severely diminished and (b) the high computational efficiency of the structured solver is preserved.

Two major classes of IB methods can be distinguished on the basis of their treatment of cut-cells. Classical IB methods use a finite volume/difference structured solver in Cartesian cells away from the irregular boundary, and discard the discretization of flow equations in the cut-cells [7]. Instead, special interpolations are used for setting the value of the dependent variables in the latter cells. Thus, strict conservation of quantities such as mass and momentum is not observed near the irregular boundary. Numerous revisions of these interpolations are still proposed for improving the accuracy and consistency of this class of IB methods [5]. A second class of IB methods (also called cut-cell methods, see e.g. [3]) aims for actually discretizing the flow equations in cut-cells. However, the calculation of fluxes in cut-cells relies usually on unstructured techniques, and their negative impact on the computational efficiency of the code is difficult to evaluate.

The purpose of this communication is to present a new IB method for incompressible viscous flows which takes the best aspect of both approaches. As the cut-cell method of Ref. [3], our method is based on the symmetry preserving finite-volume discretization on Cartesian grids by Verstappen & Veldman [10]. However, the major difference is that we have undertaken
representing the irregular boundary by its level-set function [8]. With the help of level-set
calculus tools, the fluxes in Cartesian and cut-cells are discretized in a unified fashion, ensuring
that the Cartesian structure of the stencils is preserved. Our method has the following distinctive
features:

- In contrast to standard IB methods, flow variables are actually computed near the irregular
  boundary, and not interpolated [5]. This feature is of utmost importance for computing
  the boundary layer of viscous flows. It is even more important for the viscoelastic flow
  computations we intend to perform in the future, for which the basic solver we present in
  this paper is the building block.
- The Cartesian structure of the discrete systems enables the use of efficient black box
  solvers for structured grids, resulting in a highly computationally efficient method.

In the final part of the paper, our method is validated for the circular cylinder flow in the
laminar regime, which has become the standard benchmark test for IB methods.

2 Numerical method

Let \( \Omega \) be a rectangular computational domain and \( \Gamma \) its surface. The governing equations are
the incompressible Navier-Stokes equations in integral form. In the following, we will consider
the finite-volume discretization of the continuity equation:

\[
\int_{\Gamma} \mathbf{v} \cdot \mathbf{n} \, dS = 0, \tag{1}
\]

where \( \mathbf{v} = (u, v) \) is the velocity, and of the \( u \)-momentum equation:

\[
\frac{d}{dt} \int_{\Omega} u \, dV + \int_{\Gamma} p \mathbf{e}_x \cdot \mathbf{n} \, dS - \frac{1}{Re} F_d = 0, \tag{2}
\]

where \( p \) is the pressure, \( Re \) is the Reynolds number, \( F_c \equiv \int_{\Gamma} (\mathbf{v} \cdot \mathbf{n}) u \, dS \) and \( F_d \equiv \int_{\Gamma} \nabla u \cdot \mathbf{n} \, dS \) are the convective and diffusive flux respectively.

Basic discretization on the MAC mesh

The Cartesian method on which our IB method is based is the second-order finite volume dis-
cretization of Verstappen & Veldman [10], which has the ability to preserve on non-uniform
cell distributions the conservation properties (for total mass, momentum and kinetic energy)
of the original MAC method [4]. The staggered arrangement of the discrete velocities on the
MAC mesh is illustrated in Fig. 1. The computational domain is divided in non-uniform Carte-
sian cells \( \Omega_{i,j} = [x_{i-1}, x_i] \times [y_{j-1}, y_j] \), of size \( \Delta x_i \Delta y_j \), and center \( x_{ij} = (x_i, y_j) \).

The surface of cell \( \Omega_{i,j} \) is divided in 4 elementary faces as \( \Gamma_{i,j} = \Gamma_e \cup \Gamma_w \cup \Gamma_n \cup \Gamma_s \) using
the usual compass notations. Cell \( \Omega_{i,j} \) is used as the control volume for discretizing the con-
tinuity equation (1), while the staggered control volume \( \Omega^u_{i,j} = [x_i, x_{i+1}] \times [y_{j-1}, y_j] \), of size \( \Delta x_i \Delta y_j \) with \( \Delta x_i = \frac{1}{2} \Delta x_i + \frac{1}{2} \Delta x_{i+1} \), is used for the \( u \)-momentum equation (2).

Compass notations are used for intermediate steps of the discretization of (2). For example, we
denote \( F^c_e \) the convective flux through the east face \( \Gamma^c_e \) of the staggered control volume, i.e. :

\[
F^c_e \equiv \int_{y_{j-1}}^{y_j} (\mathbf{v} \cdot \mathbf{n}_e) u(x_{i+1}, y) \, dy. \tag{3}
\]
The LS-MAC mesh for discretization in irregular domains

We consider now an irregular fluid domain $\Omega^f$ which is embedded in the computational domain $\Omega$. To keep track of the irregular boundary $\Gamma^f$, we employ a signed distance function $\phi(x)$ (i.e., the level set function [8]) such that $\phi$ is negative in the fluid region $\Omega^f$, $\phi$ is positive in the solid region $\Omega^s = \Omega \setminus \Omega^f$, and such that the boundary $\Gamma^f$ corresponds to the zero level-set of this function, i.e.:

$$
\phi(x) \equiv \begin{cases} 
-\Delta, & x \in \Omega^f, \\
0, & x \in \Gamma^f, \\
+\Delta, & x \in \Omega^s,
\end{cases}
$$

(4)

where $\Delta$ is the distance from $x$ to the closest point on the irregular boundary. This leads to the modification of the MAC mesh that is described in Fig. 2, and that will be subsequently referred to as the LS-MAC mesh. In this figure, the cell $\Omega_{i,j}$ which is part fluid / part solid, is commonly called a cut cell. The discretization of the fluxes in cut cells is performed with the help of an additional variable, denoted $\phi_{i,j}$, that stores the values of the level-set function $\phi$ at the upper right corner of the cells. The level-set values are used to efficiently compute quantities relevant to the irregular boundary and the cut-cells, such as outward normal vectors or surface integrals over $\Gamma^f$, areas and volumes of the cut-cells, etc... In this respect, the most important quantity for the LS-MAC discretization is the fluid portion of the cell faces. For example in Fig. 2, by using one-dimensional linear interpolation of $\phi(x_i, y)$ in $[y_{j-1}, y_j]$, the length of the face $\Gamma_e$ which belongs to the fluid domain is:

$$
y_n - y_{j-1} = \theta^u_{i,j} \Delta y_j, \quad \text{with} \quad \theta^u_{i,j} = \frac{\phi_{i,j-1}}{\phi_{i,j} - \phi_{i,j-1}} \quad \text{since} \quad \phi(x_i, y_n) = 0.
$$

(5)

The quantity $\theta^u_{i,j} \in [0, 1]$, which is called the cell-face ratio, shall be used to:

- Detect whether the discrete velocities in the LS-MAC mesh are in the fluid region. For example in Fig. 2, velocity $u_{i,j}$ has to be computed since $\theta^u_{i,j} > 0$, but $v_{i,j}$ should not since $\theta^v_{i,j} = 0$.
- Discretize the surface and volume integrals of the Navier-Stokes equations (1)-(2) in the cut-cells. For example, the volume of the trapezoidal cut-cell in Fig. 2 is:

$$
V_{i,j} = \frac{1}{2} \left( \theta^u_{i-1,j} + \theta^u_{i,j} \right) \Delta x_i \Delta y_j.
$$

(5)
Implement boundary conditions in the fashion of the ghost fluid method [8]. For example, the diffusive flux through the solid north boundary $\Gamma_n$ in Fig. 2 is computed as:

$$F_d^n \equiv \int_{\Gamma_n u} \partial u \partial y \, dx \approx \bar{\Delta} x_i u(x_i, y_n) - \frac{1}{2} \theta_{i,j}^u \Delta y_j,$$

with $u(x_i, y_n)$ given by the Dirichlet conditions at the irregular boundary.

These points will be discussed in greater lengths in a forthcoming paper (see also [1]). In the following, we will mainly detail the discretization of the continuity equation (1).

**Discretization of the continuity equation**

As in the Cartesian method of Verstappen & Veldman [10], the starting point of the LS-MAC discretization concerns the continuity equation (1), that reads in cell $\Omega_{i,j}$:

$$u_{i,j} - u_{i-1,j} + v_{i,j} - v_{i,j-1} = 0,$$

where mass fluxes across cell faces are denoted with a bar; for example the mass flux through the east face $\Gamma_e$ in Fig. 2 is:

$$\bar{u}_{i,j} \equiv \int_{y_{j-1}}^{y_n} u(x_i, y) \, dy.$$

In order to easily discretize this integral, we first locate the discrete unknown $u_{i,j}$ in the middle of the fluid part of the face as $u_{i,j} \equiv u(x_i, y_{j-1} + \frac{1}{2} \theta_{i,j}^u \Delta y_j)$. Then, by using midpoint quadrature, we finally discretize (8) as:

$$\bar{u}_{i,j} \approx \theta_{i,j}^u \Delta y_j u_{i,j}.$$

After similar calculations for the other cell faces, the discretization of (7) is:

$$\Delta y_j \left( \theta_{i,j}^u u_{i,j} - \theta_{i-1,j}^u u_{i-1,j} \right) + \Delta x_i \left( \theta_{i,j}^v v_{i,j} - \theta_{i,j-1}^v v_{i,j-1} \right) = 0,$$

and, in the case of a regular Cartesian cell ($\theta = 0$ or 1 only), the discrete continuity equation reduces to the one of the standard MAC discretization.
Further computational details

Discretization of the \( u \)–momentum equation (2) is performed in staggered control volumes as shown in Fig. 2. Volume integrals are discretized as:

\[
\frac{d}{dt} \int_{\Omega} u \, dV \approx V_{i,j}^n \frac{du_{i,j}}{dt},
\]

and the volume of the cut-cells is efficiently computed with the cell face ratios (see Eq. (5)). Discretization of the convective flux follows the lines of the symmetry preserving discretization of [10]. For example, Eq. (3) is discretized as \( F_c \approx \frac{1}{2} \left( \bar{u}_{i,j} + \bar{u}_{i+1,j} \right) u_e \), and central interpolation with constant weights for \( u_e \) is used. The discretization of the diffusive fluxes is more involved and is not detailed in this paper. The pressure gradient in Eq. (2) is simply computed as:

\[
\frac{1}{V_{i,j}^n} \left[ \int_{\Gamma_p^e} p \, dx - \int_{\Gamma_p^w} p \, dx \right] \approx \frac{p_{i+1,j} - p_{i,j}}{\Delta x_i^p},
\]

with \( \Delta x_i^p = \frac{1}{2} \Delta x_i + \frac{1}{2} \Delta x_{i+1} \) for the cut-cell of Fig. 2.

The time stepping method we use is the semi-implicit AB/BDF 2 projection scheme (see e.g. [2]). With our LS-MAC discretization, the pressure Poisson equation of the projection step is a symmetric linear system, as in the Cartesian case. This linear system is efficiently solved with a black-box multigrid solver for Cartesian grids.

3 Numerical results

![Figure 3: At left: geometry and boundary conditions of the flow over a circular cylinder. At right: meshes from our simulations. One mesh line out of three is shown in both direction.](image)

We have validated our method on both steady and unsteady flows around a circular cylinder in a free-stream (see Fig. 3 (left)). The Reynolds number is based on the constant inlet velocity and the cylinder diameter \( D \). For all computations, the inlet boundary is located at \( X_u = 8 \) length units upstream of the obstacle; the value of the time step is \( \Delta t = 0.01 \). We use non-uniform Cartesian meshes refined in the wake of the cylinder as shown in Fig. 3 (right), with \( N_x \times N_y \) cells. In the vicinity of the cylinder, the cell distribution is uniform of size \( h = 0.04 \). For a shape as simple as a cylinder profile with unit diameter, the level set function takes the simple analytic expression:

\[
\phi(x, y) = \frac{1}{2} - \sqrt{(x - x_c)^2 + (y - y_c)^2},
\]
Table 1: Summary of the steady computations at $Re = 20$ and comparison with results published in the literature. $L_w$ : wake length, $u_{\text{min}}$ : velocity extremum in the near wake. $C_D$ : drag coefficient.

<table>
<thead>
<tr>
<th>$N_x \times N_y$</th>
<th>$A$</th>
<th>$X_d$</th>
<th>$L_w$</th>
<th>$u_{\text{min}}$</th>
<th>$C_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1 300 $\times$ 260</td>
<td>12</td>
<td>15</td>
<td>0.952</td>
<td>-0.032</td>
<td>2.025</td>
</tr>
<tr>
<td>S2 300 $\times$ 300</td>
<td>16</td>
<td>15</td>
<td>0.956</td>
<td>-0.032</td>
<td>2.017</td>
</tr>
<tr>
<td>Experiments [11]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Linnick &amp; Fasel [6]</td>
<td>0.93</td>
<td></td>
<td></td>
<td>2.06</td>
<td></td>
</tr>
</tbody>
</table>

Table 2: Summary of the unsteady computations at $Re = 100$ and comparison with results published in the literature. $St$ : Strouhal number, $\overline{C_D}$ : time average of drag coefficient. $C_{L_{\text{max}}}$ : peak value of lift coefficient. All quantities have been computed from a time signal equal to 280 units, which gives a frequency resolution equal to $\pm 1.8 \times 10^{-5}$.

<table>
<thead>
<tr>
<th>$N_x \times N_y$</th>
<th>$A$</th>
<th>$X_d$</th>
<th>$St$</th>
<th>$\overline{C_D}$</th>
<th>$C_{L_{\text{max}}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1 300 $\times$ 260</td>
<td>12</td>
<td>15</td>
<td>0.169</td>
<td>1.341</td>
<td>0.350</td>
</tr>
<tr>
<td>S2 300 $\times$ 300</td>
<td>16</td>
<td>15</td>
<td>0.169</td>
<td>1.339</td>
<td>0.349</td>
</tr>
<tr>
<td>S3 336 $\times$ 300</td>
<td>16</td>
<td>20</td>
<td>0.169</td>
<td>1.339</td>
<td>0.349</td>
</tr>
<tr>
<td>Experiments [11]</td>
<td></td>
<td></td>
<td>0.16 - 0.17</td>
<td>1.21 - 1.41</td>
<td>-</td>
</tr>
<tr>
<td>Persillon &amp; Braza [9]</td>
<td>0.165</td>
<td></td>
<td>1.253</td>
<td>0.395</td>
<td></td>
</tr>
<tr>
<td>Linnick &amp; Fasel [6]</td>
<td>0.166</td>
<td></td>
<td>1.34</td>
<td>0.333</td>
<td></td>
</tr>
</tbody>
</table>

where $(x_C, y_C)$ is the cylinder center. This expression is discretized at cell corners prior to the time integration.

For both steady flow at $Re = 20$ and time-periodic flow at $Re = 100$, we have studied the influence of the locations of the outflow boundary $X_d$ and solid blockage $A$ (see Tables 1 and 2). For $Re = 100$ in particular, it can be observed that grid independent results are reached for $A = 16$ and $X_d = 15$. These tables also compares our results with selected experimental and numerical studies. Our computations show an overall good agreement with these results.

References