Free boundary method for calculating compressible viscous flows on unfitted meshes

Igor Menshov and Pavel Pavlukhin
Keldysh Institute for Applied Mathematics RAS, Moscow 125047, Russia
E-mail: menshov@kiam.ru

Abstract. We further develop the Free Boundary Method (FBM) for calculating compressible flow equations on Cartesian unfitted grids which has been recently proposed for the Euler model of inviscid equations. The method belongs to the class of immersed boundary methods. The effect of the solid surface is taken into account by means of the compensating flux applied on the element of solid surface in the cut cell. This flux is introduced to compensate losses in mass, momentum, and energy which occur when we virtually remove the solid element from the cut cell. The method is extended to the Navier-Stokes equations. We show how the viscous compensating flux should be introduced to assure the adequate solution in the flow domain, discuss its discrete approximation and propose the numerical method for solving the system of discrete equations. Calculation of the compensating flux requires a few data about the solid geometry in the cut cell, namely fluid volume fraction, area of the solid element and its unit normal, and baricentric coordinates of the fluid sub-element. All the data can be obtained from the level set method for setting the solid geometry. Numerical results concern testing the FBM on the solution of several benchmark viscous problems of aerodynamics.

1. Introduction

The present paper addresses a numerical technique for solving the Navier-Stokes model of non-stationary compressible viscous flow equations in domains external to solid objects on Cartesian grids unfitted to the geometry of the object. The method pertains to the class of immersed boundary methods.

In the immersed boundary approach, the computational domain (that includes both gas and solid regions) is discretized with an unfitted to solid objects grid (structured or unstructured) so that the solid surfaces may intersect computational cells. The main advantage of the immersed boundary method compared with body fitted grid approaches is the simplicity of grid generation, and the possibility to discard regridding from calculations (to accommodate changes in geometry if solid objects are moving). The penalty for this simplicity is the problem of solid surface treatment, in particular the cut cell problem – how to calculate flow parameters in the cells that are cut by the solid surface?

Since the pioneering work by C. Peskin (1977) [1] there were many efforts to cope with this problem which can be classifying into two groups. One is based on the finite volume formulation. In this approach a naïve finite volume discretization is applied to fractional cells that come out due to cutting cells of the base grid. This way faces many difficulties, such as “small cell problem”, uprising new fluid cells and collapsing cells in consequence of solid objects motion, etc. [2]. The other group of methods is based on the finite-difference discretization and employs rather sophisticated interpolation
schemes to treat the boundary conditions at solid surfaces. These methods are mostly developed for incompressible flow problems. Their generalization to compressible flows is in somewhat trickish problem due to the presence of shock waves and other discontinuities.

We develop the finite-volume method for solving compressible non-stationary Navier-Stokes equations in geometrically complex domains with using simple unfitted Cartesian grids. This method belongs to the class of immersed boundary methods when an extended region including both the fluid flow domain and the solid geometry is discretized in space with a simple, in most cases Cartesian mesh that is unfitted to the solid surface. To treat numerically the inner boundary conditions on that surface in cut cells, the method named as Free Boundary Method (FBM) has been proposed in [3-5] for the Euler model of inviscid gas dynamics equations. The method is based on the Godunov approach with the numerical flux approximated on the exact solution to the Riemann problem. The calculation is executed over the set of actual mesh cells containing both fluid and cut cells in a manner as there would be no presence of solid surface in cut cells. The effect of the solid surface is taken into account by means of the compensating flux applied on the element of solid surface in the cut cell. This flux is introduced to compensate losses in mass, momentum, and energy which occur when we virtually remove the solid element from the cut cell. It can be proven that the solution of the initial value (Cauchy) problem with the compensating flux embedded in the governing equations in the extended domain is equivalent to the solution of the conventional boundary value problem in the fluid flow subdomain. This approach is extended to the Navier-Stokes equations.

We show how the viscous compensating flux should be introduced to assure the adequate solution in the flow domain, discuss its discrete approximation, and propose the numerical method for solving the system of discrete equations. The calculation algorithm is homogeneous; all computational cells are treated in a unique manner. The only difference in computing fluid and cut cells is adding the compensating flux in summation of conventional fluxes. Calculation of the compensating flux requires a few data about the solid geometry in the cut cell, namely fluid volume fraction, area of the solid element and its unit normal, and baricentric coordinates of the fluid sub-element. All the data can be obtained from the level set method for setting the solid geometry. Numerical results concern testing the FBM on the solution of several benchmark viscous problems of aerodynamics.

2. Formulation of the FBM

The FBM method much relies on fundamental ideas of the Godunov approach. A key point of this method is an alternative mathematical formulation of the fluid dynamics problem. We consider the model of compressible viscous flow given by the system of Navier-Stokes equations. Therefore, the conventional mathematical statement is to solve the Navier-Stokes equations in the domain that is out the solid region with appropriate boundary conditions at the solid surface. Let $G$ be the solid region, and $\gamma$ is the solid surface. Then, we are seeking the solution to the system of equations

$$
\frac{\partial \mathbf{q}}{\partial t} + \frac{\partial \mathbf{f}_k}{\partial x_k} = \frac{\partial \mathbf{g}_k}{\partial x_k}, \quad \mathbf{x} \in \mathbb{R}^3 \setminus G,
$$

with the boundary conditions at surface $\gamma$ and at infinity

$$
\mathbf{u} = 0, \quad \mathbf{x} \in \gamma, \quad \mathbf{q} = \mathbf{q}_\infty, \quad \mathbf{x} \rightarrow \infty,
$$

where $\mathbf{u}$ is the velocity vector, $\mathbf{q}$, $\mathbf{f}_k$, and $\mathbf{g}_k$ are the conservative vector, the inviscid fluxes, and the viscous fluxes of the Navier-Stokes model, respectively.

The alternative formulation can be introduced in the following way. Let us suppose for a moment that all solid objects are removed and the gas occupies all the space including subdomains of solids. Then the weak form of the system of equations (1) will describe the viscous gas flow in the whole space,

$$
\frac{d}{dt} \int_V \mathbf{q} dV + \int_s \mathbf{f}_k n_k ds = \int_s \mathbf{g}_k n_k ds,
$$

(3)
where $S$ is the boundary of $V$, $V$ is a closed domain in $\mathbb{R}^3$, $n = (n_k)$ is the outward to $V$ unit normal.

The boundary conditions will not be satisfied in this case. In particular, the gas will flow through the boundary $\gamma$ transferring an amount of mass, momentum, and energy into the solid domains. Moreover, the viscous friction and the energy dissipation due to non-slip boundary conditions at the wall $\gamma$ will not be taken into account. Should restore the solid surface $\gamma$ on its proper position, the flow outside $G$ takes changing to adopt the boundary conditions on the solid surface. Our method is to model this process with effective fluxes applied at points of the solid surface $\gamma$. More precisely, we propose to modify Eq. (1) by introducing in the right-hand side additional terms – compensating fluxes as follows:

$$
\frac{d}{dt} \int_V \mathbf{q} dv + \int_S \mathbf{f}_n n_k ds = \int_S \mathbf{g}_n n_k ds - \int_{\gamma(V)} (\mathbf{f}_n n_k - \mathbf{f}_w) ds - \int_{\gamma(V)} (\mathbf{g}_n n_k - \mathbf{g}_w) ds ,
$$

where $(n_k)$ in the integrals over $\gamma(V)$ are components of the unit normal $n$ to $\gamma(V)$ directed to the gas, and $\mathbf{f}_w$, $\mathbf{g}_w$ are so-called inviscid and viscous wall reaction fluxes. These fluxes aim to account for the effect of the wall on the gas flow.

Eqs. (4) may appear to be quite similar to those used in the immersed boundary method based on penalty functions [6]. However, it is not the fact, and the two approaches are different. The penalization functions used in [6] are volumetric. The solid geometry in this method is voxel-based. In the present method, in contrast to the penalization method, the compensating fluxes are calculated on the solid surface; the method in whole employs some geometry characteristics at the subcell level, as seen below.

The modified system (4) is solved in the whole space $\mathbb{R}^3$. The wall reaction fluxes in (4) should be chosen so that the reduction of the solution of Eq. (4) in $\mathbb{R}^3 \setminus G$ exactly matches the solution of the boundary value problem (1) and (2). In fact, the flux $\mathbf{f}_w$ is defined by the wall pressure $p_w = (0, p_w n_3, 0)^T$, and the flux $\mathbf{g}_w$ is defined by the vector of local wall shear stress $\tau_w$ (skin friction), $\mathbf{g}_w = (0, \tau_w n_3, 0)^T$. The wall pressure and shear stress represent by essence the instantaneous response of the wall on the flow with the state vector $\mathbf{q}|_{k_{cy}}$. These parameters will be detailed below in the discrete model.

3. Discrete Model

The discrete model is derived on the base of the finite-volume method. The solid surface $\gamma$ is represented by the set of discrete plane elements, $\gamma = \bigcup \gamma_i$, which are of triangle or polygon shape.

With the solid surface, cells of the Cartesian base grid are classified into three groups: fluid, solid, and cut. The cut cell besides the flow state vector $\mathbf{q}$ is also characterized by the data of the sub-cell solid geometry. Here we use the linear sub-cell reconstruction which is defined by the following parameters: $\omega_f =$volume fraction of fluid, and the normal directed to the gas region $n_f$, $|n_f| = s_f$, where $s_f$ is the area of the linear element inside the cell. Details of calculating these parameters can be found in [4].

The discretization of Eqs. (4) is performed in two stages with using the principle of splitting in physics. At the first stage, the basic system of equations without terms with compensating fluxes on the l.-h.s. is integrated over all the set of fluid and cut cells as if there are no solid objects. To do this, we employ the Godunov method with the MUSCL-type subcell reconstruction which yields the following solution:
\( q_i^\ast = q_i^n - \frac{\Delta t}{\Delta V_i} \sum_{\sigma} (T_{\sigma}^{-1} F_{\sigma} - G_{\sigma}) \Delta s_{\sigma}, \)

\( F_{\sigma} = F_{\sigma} (T_{\sigma} q_{\sigma}^i, T_{\sigma} \tilde{q}_{\sigma(i)}), G_{\sigma} = (g_n n_i)_\sigma = G_{\sigma} (q_{\sigma}^i, \tilde{q}_{\sigma(i)}, \nabla q_n^i, \nabla q_{\sigma(i)}^i). \)

where \( \sigma \) indicates the face-related value, \( T_{\sigma} \) is the transformation matrix to the local face-related basis, the sub- and superscript \( \sigma \) indicates interpolated to the face centre value, the subscript \( \sigma(i) \) denotes the cell bordering the \( i \)-th cell across the interface \( \sigma \), upper tilde designates values updated to the intermediate time level properly chosen for each computational cell. The inviscid flux \( F_{\sigma} \) is approximated with the Godunov method using the exact solution of the face Riemann problem with face-interpolated data as initial data. The viscous flux \( G_{\sigma} \) is calculated with the conventional central difference scheme. The explicit scheme (5) can be replaced with the explicit–implicit time marching scheme [9], which ensures the minimum involvement of the dissipative implicit component and guarantees the maximum norm diminishing (MND) property in the case of linear equations. With a suitable choice of the time step, this scheme automatically becomes the baseline explicit second-order accurate scheme in time and space, for which the MND property holds. Details can be found in [4, 9].

We do not use ghost cells to treat neighboring solid cells in this approach. Gradients are calculated on the reduced stencil that includes only fluid and cut cells, if there are solid cells in the stencil. Computing inviscid and viscous fluxes in (5), simple extrapolation is implemented when the face separates a solid cell, i.e., \( q_{\sigma(i)} = q_i \), if the \( \sigma(i) \)-th cell is found to be solid.

The second stage serves to correct the solution of Eq. (5) with the compensating fluxes. This is done for cut cells, only. The derivation of the corresponding discrete equations can be carried out in the following way. Taking the solution \( q_i^\ast \) as an initial stage, the system of Navier-Stokes equations is integrated over the fluid part of the cut cell. This yields

\[
\omega_f \Delta V_i \frac{dq_i}{dt} = -\sum_{\sigma \in f} (T_{\sigma}^{-1} F_{\sigma} - G_{\sigma}) \Delta s_{\sigma} + (f_u + g_u) \Delta \gamma,
\]

where the summation in the r.-h.s. is done only over those faces that in whole or partly are fluid, and \( \Delta s_{\sigma} \) is the area of the fluid part of the face. As the second stage is to take into account the effect of the wall, we will approximate the first term in the r.-h.s. \( (F_{\sigma} \text{ and } G_{\sigma}) \) on the solution \( q_i^\ast \). Then, using the conservative property of these fluxes, Eq. (6) can be recast as

\[
\omega_f \Delta V_i \frac{dq_i}{dt} = -(T_{\gamma}^{-1} F_{\gamma,i} - f_u + G_{\gamma,i} - g_u) \Delta \gamma.
\]

For integrating Eq. (7) we employ an implicit time marching scheme in order to relax stiff time step restrictions due to small values of the volume fraction, which may be very small in some cells. This leads to the following discrete equations:

\[
q_i^{n+1} = q_i^\ast - \frac{\Delta \gamma \Delta t}{\omega_f \Delta V} \Phi_n (q_i^{n+1}), \quad \Phi_n = -(T_{\gamma}^{-1} F_{\gamma,i} - f_u + G_{\gamma,i} - g_u).
\]

Combining the stages, Eq. (5) and (8), come to the resulting numerical scheme:
\[ q^{n+1}_i = q^n_i - \Delta t \sum_{\sigma} \left( T_{\sigma}^{-1} F_{\sigma} - G_{\sigma} \right) \Delta s_{\sigma} - \frac{\Delta \gamma \Delta t}{\omega_{\gamma} \Delta V} \Phi_{\gamma}(q^{n+1}_i), \] (9)

\[ F_{\sigma} = F_{\sigma}(T_{\sigma} q^n_{\sigma}, T_{\sigma} q^n_{\sigma(i)}), G_{\sigma} = (g, n_i)_\sigma = G_{\sigma}(q^n_{\sigma}, q^n_{\sigma(i)}, \nabla q^n_{\sigma}, \nabla q^n_{\sigma(i)}). \]

Solution of the non-linear system of equations (9) is obtained with the matrix-free approximate factorization LU-SGS method [7]. This is a very efficient solution method; it reduces to forward and backward loops over computational cells which in fact carry out an explicit algorithm. For details of this method we refer to [4, 8].

4. Wall reaction terms

To complete the method, we need define terms \( p_w \) and \( \tau_w \) that determine the compensating fluxes. The quantity \( p_w \) represents the instantaneous pressure reaction at the solid wall to the gas flow in normal direction. This can be modelled by the piston problem. Therefore, the pressure wall reaction is defined depending on the sign of the normal relative velocity. For an ideal gas with the ratio of specific heats \( \gamma \) this reads as

\[ p_w = \begin{cases} p \left[ 1 + \frac{\gamma(\gamma+1)}{4} M^2 + \sqrt{\frac{\gamma^2 M^2 + \frac{\gamma^2 (\gamma+1)^2}{16}}{M^2}} \right], & \text{if } u_n n_k < 0, \\ p \left[ 1 - \frac{\gamma - 1}{2} M \right]^2, & \text{otherwise} \end{cases} \] (10)

where \( M = u_n n_k / a \) is the Mach number of the flow in the normal direction to the wall, \( a \) is the speed of sound.

The viscous wall reaction term \( \tau_w \) is, by essence, the local wall shear stress, \( \tau_w = \sigma_{\gamma} n \), where \( \sigma \) denotes the stress tensor, and the subscript \( \gamma \) indicates that the tensor is taken at a point of the solid surface. Approximating shear stress \( \sigma_{\gamma} \), we assume a linear distribution of the velocity vector in the normal direction so that the normal derivative is evaluated as

\[ \frac{\partial u}{\partial n} = \frac{u}{h_n}, \] (11)

where \( h_n \) is a characteristic length that is adopted in this paper as the distance from the barycenter of the fluid part of the cut cell to the solid geometry \( \gamma \). \( h_n = \rho(x_b^c, \gamma) \). Another simple option with lesser data of the geometry in the cut cell is to take \( \alpha \omega_{\gamma} \Delta V / \Delta \gamma \) as the characteristic length in the cut cell with the scaling factor \( \alpha \sim 1/3 - 1/2 \). The numerical results presented below have been obtained with the first option.

The other two derivatives in tangential to the solid interface directions are calculated by using the cell gradient tensor of the velocity vector \( \nabla \mu_i \), which is approximated with the LSM on the stencil consisting of fluid and cut cells, only. Given this data, the viscous wall reaction can be written after some manipulations as

\[ \tau_w = \sigma_{\gamma} n = \frac{\mu}{h_n} u + \frac{\mu}{3} \left( \frac{u_n}{h_n} - n_i n_j \nabla \mu_i - 2 \nabla \mu_i \right) n + \mu \eta \nabla u_i, \] (12)

and the viscous compensating flux in (8) is written as
\[
G_{c,x} - g_n = \mu \left( \frac{u}{h_w} - n_j n_j \nabla_j u \right) + \frac{1}{3} \mu \left( \frac{u}{h_w} - n_j n_j \nabla_j u \right) n .
\]  (13)

5. Numerical results

The method was first tested on the inviscid flow past the NACA0012 airfoil. In this calculations we neglect the viscous effects, \( \sigma = \tau_w = 0 \). The angle of attack was specified as \( \alpha = 1.25^\circ \), and the free-stream Mach number was \( M = 0.8 \). Meshes of two types were used: a body-fitted structured C-grid with 400 cells along the airfoil boundary and with a general resolution of \( 1000 \times 150 \) cells and a body-unfitted Cartesian grid with a resolution of \( 200 \times 24 \) inside a rectangle circumscribed about the airfoil and with a general resolution of \( 650 \times 324 \).

The computations on these grids are performed using the hybrid explicit–implicit second-order accurate scheme with the Courant number \( C = 10 \). Figure 1 shows the distribution of the pressure coefficient \( C_p \) and the convergence rates to the steady-state solution. It can be seen the results obtained on different grids, including the convergence rate, agree well. A characteristic feature of the flow is a small local minimum in the pressure distribution arising behind the shock wave on the upstream side of the airfoil.

Evidently, this feature is better reproduced on the Cartesian than body-fitted grid. In fact, the local minimum is absent on the latter grid and appears as the grid resolution is increased. This finding is associated with the orthogonality property, which is inherent in the Cartesian grid and ensures higher accuracy of the scheme. Below are the lift and drag coefficients, \( C_l \) and \( C_d \), computed on the body-fitted and body-unfitted Cartesian grids for NACA0012, \( M = 0.8 \), \( \alpha = 1.25^\circ \). The difference between their values is less than 1%.

| Coefficient | Cartesian grid | C-grid |
|-------------|----------------|--------|
| \( C_l \)   | 0.3012         | 0.3036 |
| \( C_d \)   | 0.02184        | 0.02199|

Next verification test is the laminar boundary layer on a flat plate at zero incidence. The inflow Mach number is 0.3, and the Reynolds number is \( 400000/\text{1m} \). The calculation is performed with two Cartesian grid. One is fitted to the plane. The computational domain is represented by a quadrangle \( (x,y) = [-0.5,1.0] \times [-0.5 \times 0.5] \). The grid consists of 300X700 cells clustering to the plate so that the
minimal size of the cell at the plate is $2.5 \times 10^{-4}$. The plate is located at $Y=0$, with the part $Y<0$ being treated inviscid wall (slip b.c.) and $Y>0$ being considered as conventional viscous wall (non-slip b.c.).

Non-fitted Cartesian grid is obtained by rotating the above computational domain and grid by a small angle $\pi / 60$. In this case the plate will cut some computational cells; non-cut cell that are below the plate are considered as solid and do not calculated.

![Figure 2. Laminar boundary layer along a plate: Self-similar profiles of the velocity component along the plate (a), enlarged region (b).](image)

In Fig. 2 we show results of this calculation. Here we mark the tangential velocity normalized with the inflow velocity vs the normalized distance to the wall $\eta = \frac{\sqrt{\rho_{\text{ref}} u_{\text{ref}}}}{\mu x}$. These data is plotted for all fluid and cut computational cells in the domain $0.1<X$. As can be seen from the figure, all the points are collapsed on a single curve that represents the analytical Blasius solution. The latter is shown in blue. Red markers correspond to the option where the characteristic length $h_w$ is taken as the distance from the barycentre of the fluid part of the cut cell to the plate. Black markers indicate the result obtained with the alternative simple option $h_w = 0.5 \frac{\omega_x \Delta V}{\Delta Y}$. The results of the calculation with the conventional body-fitted grid are not shown in Figure as these results nearly coincide the red markers. From these data one can conclude that the characteristic length is not so critical for the accuracy of the FBM; a simple estimation of this parameter gives slightly overpredicted values as seen in Fig. 2b.

6. Conclusions
The free boundary method has been extended to the Navier-Stokes equations. The method is capable to solve problems of compressible viscous flows around solid objects on simple unfitted Cartesian grids. To treat viscous non-slip boundary conditions on solid surfaces, the viscous compensating flux is added to the inviscid compensating flux in the r.-h.s. of the governing equations. A discrete analogue of the compensating flux has been proposed. The numerical method to solve the resulting modified system of equations has been discussed. The method is verified on two typical problems of aerodynamics: inviscid flow past an airfoil NACA0012 and viscous laminar boundary layer on a flat plate. Unfitted grid calculations with the proposed free boundary method have been compared with standard body fitted grid calculations. This analysis has shown capability of the method to accurately model inner viscous boundary conditions on unfitted grids by means of properly defined compensating fluxes.

Acknowledgments
This research was supported by the Grant No 17-71-30014 from the Russian Science Foundation.
References

[1] Peskin C S 1977 Numerical analysis of blood flow in the heart *J. Comput. Phys.* **25** 220

[2] Pember R B, Bell J B, Colella P, Crutchfield W Y and Welcome M L 1995 An adaptive Cartesian grid method for unsteady compressible flow in irregular regions *J. Comput. Phys.* **120** 278

[3] Menshov I S and Kornev M A 2014 Free-Boundary Method for the Numerical Solution of Gas-Dynamic Equations in Domains with Varying Geometry *Mathematical Models and Computer Simulations* **6** 612

[4] Menshov I S and Pavlukhin P V 2016 Efficient Parallel Shock-Capturing Method for Aerodynamics Simulations on Body-Unfitted Cartesian Grids *Computational Mathematics and Mathematical Physics* **56** 1651

[5] Menshov I and Pavlukhin P 2017 Highly scalable implementation of an implicit matrix-free solver for gas dynamics on GPU-accelerated clusters *J Supercomput* **73** 631

[6] Boiron O Chiavassa G and Donat R 2009 A high-resolution penalization method for large Mach number flows in the presence of obstacles *Computers & Fluids* **38** 703

[7] Jameson A and Turkel E 1981 Implicit schemes and LU decomposition *Math. of Comp.* **37** 385

[8] Menshov I and Nakamura Y 1995 An implicit advection upwind splitting scheme for hypersonic air flows in thermochemical nonequilibrium *Collection of technical papers of 6th Int. Symp. on CFD, Lake Tahoe, Nevada* 815

[9] Menshov I and Nakamura Y 2004 Hybrid explicit-implicit, unconditionally stable scheme for unsteady compressible flows *AIAA J.* **42** 551