Adaptive Mesh Refinement Strategies for Immersed Boundary Methods

E. Balaras* and M. Vanella†

University of Maryland, College Park, MD, 20742, USA

In the present study we propose an adaptive mesh refinement (AMR) strategy which can be coupled with embedded boundary methods in a straightforward manner. The building-block of the AMR solver is a structured Cartesian solver employed on a hierarchy of sub-grids. A fractional-step formulation is used for the time advancement, and all spatial derivatives are discretized on a staggered grid using second-order, central finite-differences. The boundary conditions on an arbitrary body immersed in the AMR grid are imposed using a direct-forcing, embedded boundary formulation. The overall method is second order accurate both in space and time. The accuracy and robustness of the approach are demonstrated for the flow around a sphere and a flow around a sphere bouncing off a wall.

I. Introduction

Recently non-boundary conforming formulations have been emerging as a viable alternative to unstructured methods for complex geometries and fluid-structure interaction problems. To compute the flow around a complex body the equations of motion are usually solved on a structured grid, which is almost never aligned with the body. Depending on the specifics of the formulation, boundary conditions can be imposed by appropriately modifying the stencil in the neighborhood of the body, or using a forcing function which can be derived either using physical arguments, or directly from the discrete problem. Although such methods have been traditionally applied to low Reynolds number flows, recently applications on complex turbulent and transitional flows with moving boundaries have also been reported with very good results. A drawback of these formulations, however, is the lack of flexibility to selectively distribute points in areas of high velocity gradients without increasing the grid resolution in all areas of the computational box. Some form of local grid refinement strategy is necessary to extend these methods to a broader range of applications.

Two general categories of adaptive refinement methods (AMR) for Cartesian grids can be identified: i) isotropic splitting of individual cells that can be managed using hierarchical trees, or fully unstructured data-structures; ii) grid embedding where block-structured grids composed of nested rectangular patches are used. The latter approach was initially introduced by Berger and Oliger for the solution of one- and two-dimensional hyperbolic problems, where the sharp discontinuities in the solution were better captured with increasingly refined rectangular grid patches. The method has been extended to three-dimensions, and proved to be a robust, cost effective approach for a range of hyperbolic problems. Applications to incompressible flows, however, have been limited primarily due to complications associated to the enforcement of the divergence-free constraint. Most algorithms for incompressible flows are based on the extension of the second-order projection method by Bell et. al. on the grid topology proposed by Berger and Oliger. In this particular formulation the viscous and advective terms in the momentum equation are advanced via a Crank-Nicholson scheme, and an unsplit, second-order upwind Godunov method is used to evaluate the nonlinear term at the time-centered location. In most cases time refinement (i.e. different time-steps are used in different AMR levels) is also employed and synchronization of the solution between AMR levels is required to ensure the divergence free constraint. Recently, applications to multiphase flows and to prototypical laminar flows have been reported.

In the present study we propose an AMR strategy which can be coupled with embedded boundary methods in a straightforward manner. The building-block of the AMR solver is a structured Cartesian solver.
employed on a hierarchy of sub-grids. A fractional-step formulation is used for the time advancement, and all spatial derivatives are discretized on staggered grid using second-order, central finite-differences. In the following sections the details of the formulation together with a few example applications will be given.

II. Methodologies

In the proposed AMR strategy a single-block Cartesian grid solver is employed on a hierarchy of sub-grids with varying spatial resolution. Each of these sub-grid blocks has a structured Cartesian topology, and is part of a tree data structure that covers the entire computational domain. The Paramesh toolkit\cite{16} is utilized for the implementation of the adaptive refinement process. The package creates and maintains the hierarchy of sub-grid blocks, with each block containing a fixed number of grid points. Local refinement of a sub-grid block is performed by bisection in each coordinate direction. An example of the grid hierarchy and nested sub-grid block refinement is shown in figure 1, where a single $4 \times 4$ block covers the computational domain at refinement level 1. Refinement of block 1A creates four sub-grid blocks at refinement level 2. Further refinement of blocks 2A and 2B creates eight sub-grid blocks at level 3, and subsequent refinement of blocks 3E and 3F creates eight sub-grid blocks at level 4. At the end of this process the computational domain is composed of sub-grid blocks at three different refinement levels (2 blocks at level 2, 6 blocks at level 3, and 8 blocks at level 4). Note that the refinement or de-refinement of the sub-grid blocks is constrained to allow a jump of no more than one refinement level at the interface of two sub-grid blocks.

Boundary conditions are required at the physical boundaries of the domain as well as the interior boundaries of the sub-grid blocks. These boundary conditions are enforced utilizing a layer of ghost cells surrounding each sub-grid block. The physical boundary conditions are enforced on the sub-grid block faces that are located on the boundary of the computational domain. The ghost cells on the interior sub-grid boundaries are filled using the solution data from neighboring sub-grid blocks. The interpolation and ghost cell filling process is illustrated in two-dimensions for cell-centered grid points on the adjacent sub-grid blocks with different refinement levels shown in Figure 2a. First, the fine grid is restricted to its parent grid, which has the same level of refinement as the adjacent coarse block by a sequence of one-dimensional quadratic interpolations using nine interior grid points on the fine-grid side (see Figure 2b-top). The interpolated values are used to fill the ghost cells for the coarse grid. Next, the ghost cells for the fine grid are filled using a similar interpolation strategy utilizing the parent grid points, including the ghost cell points from the adjacent coarse-grid block (see Figure 2b-bottom).

Both advective and viscous terms in the Navier-Stokes equations for incompressible flow are advanced in time using an explicit Adams-Bashforth fractional step formulation. The fully explicit formulation enables an efficient computation of the predictor and corrector steps by utilizing the ghost cells at the boundaries between blocks. A multigrid approach is utilized to solve the elliptic pressure Poisson equation. This is a natural choice considering the hierarchy of sub-grid blocks covering the computational domain. The algorithm uses full multigrid cycles, in which all relaxation sweeps extend across the full computational domain. To illustrate this process, consider the domain shown in Figure 1d. A relaxation sweep is performed over the full domain composed of sub-grid blocks at three different refinement levels (2 blocks at level 2, 6 blocks at level 3, and 8 blocks at level 4). When restricting to the next coarser refinement level, only the
sub-grid blocks at the finest level undergo the restriction operation. Therefore, the eight blocks at level 4 in Figure 1d are restricted to two blocks at level 3, and the next relaxation sweep is performed over the domain shown in Figure 1c. After reaching the coarsest grid (Figure 1a), relaxation sweeps are followed by prolongation of the solution to the next higher refinement level. The relaxation sweeps are carried out by looping over each sub-grid block, which requires each block’s ghost cell data to be updated after each sweep. Within the multigrid solver, linear interpolation is used for prolongation, restriction, and ghost cell filling, which aids in computational efficiency and convergence speed without sacrificing second-order accuracy of the solution.

The boundary conditions on an arbitrary body immersed in the AMR grid are imposed using a direct-forcing, embedded boundary formulation. We have developed a scheme which is very robust and efficient when dealing with multi-body fluid-structure interactions. The main feature of this direct forcing approach is that the forcing function is computed on the Lagrangian markers rather than the Eulerian points unlike the classical direct forcing schemes. The basic process is demonstrated in Figure 2c, where: i) for each Lagrangian marker on the surface of the body the closest point on the Eulerian grid is found; ii) a support-domain around each Lagrangian marker is defined, in which the shape functions based on a moving least squares formulation will be constructed; iii) the transfer functions that will be moving information from the Lagrangian to the Eulerian grid and vice-versa are build. The method gives comparable results to boundary conforming formulations.

III. Numerical Examples

III.A. Flow around a sphere at Re=300

The flow around a sphere at $Re = U D / \nu = 300$, where $U$ is the freestream velocity, $D$, the diameter of the sphere and $\nu$ the kinematic viscosity of the fluid, has been computed. The flow domain is $40D \times 20D \times 20D$ in the streamwise, $(x)$, cross-stream, $(z)$, and spanwise, $(y)$, directions respectively, and the sphere is located $10D$ downstream of the inflow plane. At the outflow plane a convective boundary condition is used, while slip conditions are applied to all other boundaries, except for the inflow plane where the velocity is set equal to the freestream velocity, $U$.

In Figure 3a the block arrangement in the $x-y$ plane is shown. A total of five refinement levels are
Figure 3. (a) Computational domain for flow around a sphere at \( Re = 300 \). A slice of the domain at \( y = 0 \) is shown, each square represents a leaf 16\(^3\) block. The maximum refinement was kept up to 2.5\( D \) downstream of the sphere. (b) Isosurface of \( Q \) contour colored by values of streamwise vorticity. The scale of vorticity ranges from -1 (blue) to +1 (red) using 40 contours. The time corresponding to this snapshot is \( t = 180.96 \).

used, and the basic block size was set to 16\(^3\) cells. The resolution near the sphere was 0.0195\( D \), placing approximately 6-7 points within the boundary layers on the surface. Overall 6.4 \times 10^6 points were used, and the grid was not adapted in time.

The temporal evolution of the drag and lateral force coefficients,

\[
C_D = \frac{F_x}{\frac{1}{2} \rho U^2 A_S}, \quad C_L = \frac{\sqrt{F_x^2 + F_y^2}}{\frac{1}{2} \rho U^2 A_S},
\]

where \( A_S = \pi D^2 / 4 \) is the sphere’s projected area, is given in Figure 4a. The value of the resulting Strouhal number is \( St = 0.132 \), which is in excellent agreement with the value 0.137 reported by Johnson and Patel\(^{19}\) for the same problem. The mean drag coefficient is \( C_{Dmean} = 0.634 \) and the amplitude of \( C_D \) oscillation is 0.0039. This agrees well with the corresponding values 0.656 and 0.0035 in the computations by Johnson and Patel.\(^{19}\) Both computations also agree well with the experimental results by Roos and Willmarth,\(^{20}\) where \( C_{Dmean} = 0.629 \) and \( C_{Lmean} = 0.0594 \) were reported.

In Figure 4b the mean and rms values of the streamwise velocity component as a function of the distance from the sphere’s center, along the symmetry axis are shown. The results by Johnson and Patel,\(^{19}\) are also shown for comparison. Our statistics were computed over the last four periods of the calculation. As it can be seen \( u_{mean} \) agrees well with the reference computation. The negative values of \( u_{mean} \) behind the sphere center are due to the presence of the recirculation zone, which is captured very accurately. Near the sphere \( u_{rms} \) also agrees well with the reference data. At streamwise distances larger than 4\( D \), however, small discrepancies can be observed due to the drastic derefinement of the AMR grid.

In Figure 4b an instantaneous snapshot of the flow at \( t = 180.96 \) is shown. The large coherent structures in the wake of the sphere are visualized by an isosurface of the second invariant of the velocity gradient tensor, \( Q = 0.01 \), which is colored by contours of streamwise vorticity. The sequence of hairpin type vortices been shed behind the sphere, typical for this Reynolds number can be seen.

III.B. Sphere-wall collision

To investigate the robustness and accuracy of the proposed method in more challenging fluid-structure interaction problems we performed computations of a rigid sphere bouncing off a wall. The dominant parameters in the collision process are the Stokes number, \( St = 1/9(\rho_b/\rho_f)Re \), where \( \rho_b \) and \( \rho_f \) are the particle and fluid densities respectively, and the Reynolds number \( Re \). \( Re \) is based on the particle diameter, \( D \), and the translational velocity, \( U_f \), an instant before impact. For low values of the Stokes number (\( St < 10 \)) no rebound will occur, even if the dry restitution coefficient, \( e_{dry} \), is different from zero. For \( St > 10 \) rebound occurs, where the total restitution coefficient, \( e_T \), is lower than \( e_{dry} \). For large values of the Stokes number (\( St > 500 \)) the total restitution coefficient, \( e_T \), approaches \( e_{dry} \) at low \( Re \). If \( e_{dry} = 0 \), then the Reynolds number is the only dominant parameter.

In the following we will focus on cases where the sphere rebounds off the wall, which are very challenging problems for any numerical method. We considered two cases, where the the Stokes number was kept constant, \( St \sim 300 \), and the \( Re \) number varied by a factor of 10. In Figure 5a a sketch of the computational
domain is shown. Both the sphere and the wall are immersed into the AMR grid and contain $2.7 \times 10^5$ and $2.9 \times 10^5$ Lagrangian markers respectively (see figure 5b). We keep the maximum refinement around the moving sphere and floor during the simulation. The resulting grid size was $\Delta_{min} = 0.015D$ for the $Re = 76.8$ computations, and $\Delta_{min} = 0.010D$ for the $Re = 830$ case. Overall, grids with an average number of $10 \times 10^6$ and $15 \times 10^6$ grid points were used respectively. The grid was also refined/derefinied in time to optimally distribute points in the domain as it can be seen in Figure 5b.

The horizontal displacements and rotations of the sphere are constrained and the vertical displacement, $z_s(t)$, is governed by:

$$m_s \ddot{z}_s(t) = -m_s g + f_z(z_s, \dot{z}_s, t),$$

(2)

where $m_s$ is the mass of the sphere, $g$ is the acceleration of gravity and $f_z$ is the hydrodynamic force on the sphere in the vertical direction. The Navier-Stokes equations governing the dynamics of the fluid, and equation (2) governing the dynamics of the sphere are solved as a coupled system using the predictor-corrector strategy proposed in Yang et al.\textsuperscript{21}
Initially the sphere is located at a distance of 4.3D from the horizontal wall, and is started from rest in the low \(Re\) case. This initial condition together with a density ratio of \(\rho_s/\rho_f = 35\) results in a Reynolds number just before impact of \(Re = 76.8\) and a Stokes number of \(St = 299\). On the higher \(Re\) calculation, the sphere is impulsively started to reach a velocity corresponding to an initial Reynolds number of \(Re_i = 510\), reaching a \(Re = 830\) and \(St = 295\) before impact. In this case the density ratio was set to \(\rho_s/\rho_f = 3.2\). The dry restitution coefficient all our computations is the one used in the axisymmetric calculations of Ardekani and Rangel, \(^{22}\) namely \(e_{dry} = 0.97\). As the sphere approaches the wall, the fluid layer height is reduced up to a distance equal to the surface roughness \(h_r\), where the bounce is assumed to occur. We assume Newton’s law of impact at this point and define a new set of initial conditions for equation (2):

\[
\begin{align*}
&z_{s2} = z_{s1}; \\
&\dot{z}_{s2} = -e_{dry}\dot{z}_{s1}
\end{align*}
\]

where \(z_{s1}, \dot{z}_{s1}\) and \(z_{s2}, \dot{z}_{s2}\) are the sphere’s vertical position and velocity before and after the impact respectively. In numerical simulations of bouncing problems, it is important to properly resolve the lubrication layer between the two bodies. With the present grid resolution and a value of the surface roughness of \(h_r = 0.05D\), a minimum of 5 Eulerian cells was always present between bodies during impact.

In figures 6a,b the position and velocity of the sphere’s center as a function of time can be seen for \(Re = 78.6\). The vertical fluid force modifies the linear behavior of the sphere’s velocity in vacuum, especially after impact. The temporal evolution of vertical force \(f_z(t)\) is also shown in figure 6c. Instants before the collision, an increase in pressure in the fluid layer between sphere and the floor produces an increase in vertical force, and forces the fluid to be ejected sideways. Once the impact occurs the velocity changes sign.
according to equation (3).

In figure 6d a more detailed view of the sphere’s velocity just after the impact can be seen for both the low and high Reynolds numbers. The fluid force reduces the sphere’s rebound velocity over a short period of time. This effect is specially pronounced on the $Re = 830$ case, where the density ratio $\rho_s/\rho_f$ is one order of magnitude lower than the one in the low $Re$ case at the same Stokes number. As a result the system’s motion at $Re = 76.8$ is mainly driven by the sphere’s inertia and gravity, while at the higher Reynolds number the fluid forces play an important part. The total coefficient of restitution $e_T$,\(^22\) is defined as the velocity of the sphere at time, $t = t^* U_f/D \simeq 0.07$, after the impact time, $t_c$, divided by the pre-collision velocity $U_f$. It measures the dissipative effect of the fluid, as it is drained, and subsequently re-enters the layer between the sphere and the wall. In our low Reynolds number case $e_T = 0.915$, which is in very good agreement with the result from the reference computations,\(^22\) where $e_T = 0.92$. For the $Re = 830$ calculation the computed effective coefficient of restitution was $e_T = 0.63$. This result follows the trend reported by Ardekani and Rangel,\(^22\) where a decrease of $e_T$ with Reynolds number was observed at constant Stokes number. In particular they found a decrease of approximately 5% as the Reynolds number increased from 35 to 162 at $St = 301$.

In Figure 7 instantaneous vorticity isolines are shown at a plane through the symmetry axis. Each plot consists of two parts: left and right, corresponding to the low and high Reynolds numbers respectively. An instant before impact, the boundary layers on the sphere’s surface and the shear layers in the wake are visible for both Reynolds numbers (see Figure 7a). In the high Reynolds number case a small recirculation area is also present. At instant in time just after the bounce (see Figure 7b) for both Reynolds numbers secondary vorticity is generated on the surface of the sphere as the primary wake vortex moves towards the wall. In the higher Reynolds number case, however, the secondary vorticity rolls into a distinct vortex and a new separation region on the other side of the sphere develops (see Figure 7c). In the low Reynolds number case the wake vorticity is dissipated and no separation region is seen in the back of the sphere. As the sphere falls back towards the wall for its second impact, the surface vorticity changes sign and a secondary vortex pinches-off (see Figure 7d). In the low Reynolds number case the sphere is much heavier and $U_f$ is higher, leading to a higher rebound distance. By the time, however, the sphere approaches maximum height, the vorticity in the wake is dissipated. For high number case on the other hand, the vortices that pinch-off during each bounce form pairs that travel along the wall.

Figure 7. Impact of a sphere against a wall. The left half on the figures corresponds to the $Re = 76.8$ result, while the right half corresponds to $Re = 830$. Vorticity normal to paper plane ranges from $-4$ (blue) to 4 (red) in $\Delta \omega_\theta = 0.116$ increments. (a) $-0.4 \tau_f$, (b) $0.08 \tau_f$, (c) $1 \tau_f$, (d) $2.8 \tau_f$, (e) $3.3 \tau_f$ for $Re = 830$ and $9 \tau_f$ for $Re = 76.8$, (f) $3.9 \tau_f$ for high $Re$ and $10 \tau_f$ for low $Re$. Here $\tau_f = D/U_f$. 

American Institute of Aeronautics and Astronautics
IV. Conclusion

Non-boundary conforming formulations have been emerging as a viable alternative to unstructured methods for complex geometries and fluid-structure interaction problems. In most cases, however, the equations of motion are solved on structured grids, and there is a lack of flexibility to selectively distribute points in areas of high velocity gradients without increasing the grid resolution in all areas of the computational box. This limits the applicability of such methods to low Reynolds numbers especially in cases with moving boundaries.

To alleviate this problem we proposed an AMR strategy, where a single-block Cartesian grid solver is employed on a hierarchy of sub-grids with varying spatial resolution. Each of these sub-grid blocks has a structured Cartesian topology, and is part of a tree data structure that covers the entire computational domain. The Paramesh toolkit\textsuperscript{16} is utilized for the implementation of the adaptive refinement process. The AMR algorithm is implemented in a fractional step formulation, where both advective and viscous terms in the Navier-Stokes equations for incompressible flow are advanced in time using an explicit Adams-Bashforth scheme. A highly efficient multigrid approach is utilized to solve the elliptic pressure Poisson equation, which is a natural choice considering the hierarchy of sub-grid blocks covering the computational domain. The accuracy and robustness of the approach have been demonstrated for the flow around a sphere, and a flow around a sphere bouncing-off a wall.

Acknowledgments

This research was supported by the Office of Naval Research under Grant No. N00014-06-1-0460, monitored by Dr. Ronald D. Joslin, and by the National Science Foundation, under Grant No. CTS-0347011.

References


