8th. World Congress on Computational Mechanics (WCCM8)
5th European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 –July 5, 2008
Venice, Italy

AN EMBEDDED BOUNDARY METHOD WITH ADAPTIVE MESH REFINEMENTS

Marcos Vanella and Elias Balaras

Department of Mechanical Engineering
University of Maryland, College Park, MD, 20742, USA
mvanella@umd.edu, balaras@umd.edu

Key Words: *Immersed Boundary Method, Finite Differences, Adaptive Mesh Refiniment, Large-Eddy Simulation, Fluid-Structure Interaction.*

ABSTRACT

In recent years non-boundary conforming formulations have been emerging as a viable alternative to unstructured grid methods especially for fluid-structure interaction problems. To compute the flow around a complex body the equations of motion are usually solved on a fixed 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 neighbourhood of the body or using a forcing function which can derived either using physical arguments or directly from the descrite problem. A limitation of such formulations, however, which are usually associated with structured Cartesian grids, is that the fine grid resolution required to capture the thin boundary layers developing on the surface of a solid body is unavoidably extended throughout the computational domain resulting in a large number of 'wasted' grid points.

In the present study we will propose an adaptive mesh refinement (AMR) strategy that can drastically reduce the total amount of grid points in such computations without compromising overall accuracy and efficiency. In particular, the fluid flow equations are discretized on a hierarchy of finite difference blocks which can be adaptively refined using successive bisections in all the coordinate directions. Each of these blocks has a structured Cartesian topology, and is part of a tree data structure that covers the entire computational domain. The Paramesh toolkit [1] is used to create and maintain the above data structure. The refinement and/or de-refinement of the sub-grid blocks is constrained to allow a jump of no more than one level at the interface. Continuity of the solution between sub-blocks at different levels of refinement is enforced using ghost cells. In Figure 1, for example, the arrangement of the grid and corresponding ghost cells at the interface between two blocks at different refinement levels is shown. The velocity and pressure at these points is interpolated from the neighbouring blocks using Lagrange polynomials. The projection scheme is also modified at the interface to conserve mass.

The presence of complex moving bodies immersed in the Cartesian sub-blocks is taken into account using an embedded boundary technique that utilizes 'direct-forcing', which is directly defined in the discretized space [2]. Following the ideas presented in [3], the direct forcing is evaluated the Lagrangian

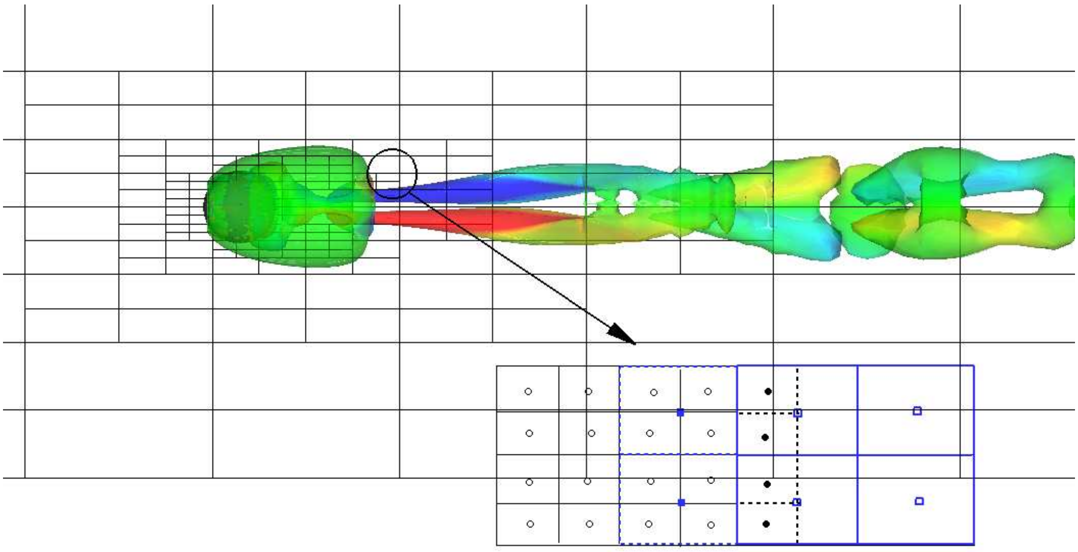

Figure 1: The flow around a smooth sphere at Re=300. Isosurfaces of the second invariant of the velocity gradient tensor are shown at an instant in time and are coloured with the magnitude of the streamwise vorticity. The block boundaries at a plane through the center of the sphere are also shown. The insert at the bottom part of the figure shows schematic of the ghost cell (filled symbols) arrangement near an interface between two block at different refinement levels.

marker points on the body and then transferred on the Eulerian grid using a moving least squares (MLS) approximation technique. For all fluid structure interaction problems that will be presented the strong coupling formulation in [4] is used. To demonstrate the accuracy and efficiency of the method we will present results in a variety of flows with increasing complexity: i) direct and large-eddy simulations of the flow around a smooth sphere at Reynolds numbers up to 10,000; ii) the flow around a realistic model of a small insect during hovering.

REFERENCES

[1] P. MacNeice, K.M. Olson, C. Mobarry, R. deFainchtein, and C. Packer. “Paramesh: a parallel adaptive mesh refinement community toolkit”. *Comput. Phys. Commun.*, Vol. **126**, 330–354, 2000.

[2] E. A. Fadlun, R. Verzicco, P. Orlandi, and J. Mohd-Yusof. “Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations”. *J. Comput. Physics*, Vol. **161**, 35–60, 2000.

[3] M. Uhlmann. “An immersed boundary method with direct forcing for the simulation of particulate flows”. *J. Comput. Phys.*, Vol. **209**, 448–476, 2005.

[4] J. Yang, S. Preidikman, and E. Balaras. “A strongly-coupled, embedded-boundary method for fluid-structure interactions of elastically mounted rigid bodies”. *J. Fluids and Structures*, In Press, 2007.