Solving Fluid Structure Interaction Problems with an Immersed Boundary Method

Michael F. Barad[1], Christoph Brehm[2], and Cetin C. Kiris[1]

[1] Computational Aerosciences Branch, NASA Ames Research Center, Moffett Field, CA, USA
[2] Department of Aerospace and Mechanical Engineering, University of Arizona, Tucson, AZ, USA

Corresponding author: michael.f.barad@nasa.gov

Abstract: An immersed boundary method for the compressible Navier-Stokes equations can be used for moving boundary problems as well as fully coupled fluid-structure interaction is presented. The underlying Cartesian immersed boundary method of the Launch Ascent and Vehicle Aerodynamics (LAVA) framework, based on the locally stabilized immersed boundary method previously presented by the authors, is extended to account for unsteady boundary motion and coupled to linear and geometrically nonlinear structural finite element solvers. The approach is validated for moving boundary problems with prescribed body motion and fully coupled fluid structure interaction problems.

Keywords: Immersed Boundary Method, Higher-Order Finite Difference Method, Fluid Structure Interaction.

1 Introduction

Immersed Boundary Methods (IBMs) have been developed for many years and have appeared in various forms since they were first introduced by Peskin[1, 2] (see for example Goldstein *et al.*[3], LeVeque and Li[4], Wiegmann and Bube[5], Linnick and Fasel[6], Johansen and Collela[7], Mittal and Iaccarino[8], Zhong[9], Duan *et al.*[10] and many others). These methods were first introduced as a nontraditional approach for numerically solving initial/boundary-value problems for complex geometries on Cartesian meshes and have matured to become increasingly important for a wide range of applications[11, 12, 13]. One of the key advantages of immersed boundary methods is that the computational meshes can be automatically generated starting from a water tight surface triangulation independent of the complexity of the geometry. For highly complex geometries, which are relevant for many fields of science and engineering, the process of generating a high-quality grid is extremely time consuming. For flows containing moving boundaries or fluid structure interaction (FSI), immersed boundary methods can dramatically simplify the numerical solution procedure by solving the governing equations on stationary, non-deforming Cartesian grids. In contrast, conventional body-fitted grid approaches are confronted with immense difficulties in the solution procedure, especially for fluid-structure interaction problems involving large relative boundary motions/deformations, in which the grids must be deformed and may become tangled.

2 Scope of Work

The final paper will discuss all aspects of coupling the high-order accurate compressible Navier-Stokes solver with a structural linear and geometrically nonlinear structural Finite Element Solver (FEM) solver. The development of these FSI capabilities are part of the overall Launch Ascent and Vehicle Aerodynamics (LAVA) framework[11] that is being developed by the authors at NASA Ames Research center.
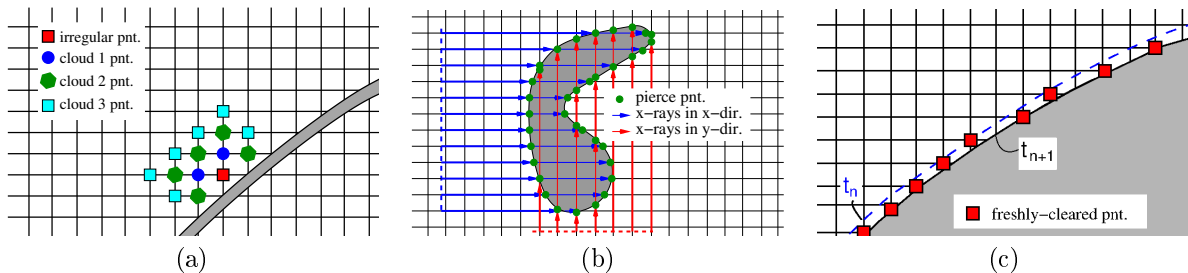
Figure 1: (a) Cloud selection process for the boundary extrapolation stencil, (b) illustration of x-ray algorithm used for in-out testing and (c) freshly cleared cells after moving the immersed boundary in the time interval $[t_n, t_{n+1}]$.

Some of the key challenges for simulating FSI problems with an immersed boundary method is the treatment of thin geometries, accounting for the thin boundary layer near the geometry, and dealing with the moving boundary. The treatment of the thin geometry affects the cloud selection process for the boundary condition extrapolation stencil since the current sharp immersed boundary method does not utilize ghost cells. The algorithm ensures that points are only selected for the point cloud if they reside on the same side of the boundary as the irregular grid point (i.e. points adjoining the boundary). Figure 1a schematically illustrates the cloud selection process at an irregular grid point. First, all irregular grid points compute their base clouds while the base clouds for the regular grid points are assumed to contain the direct neighbors, i.e., $(i \pm 1, j, k)$, $(i, j \pm 1, k)$, and $(i, j, k \pm 1)$. Then the point cloud at the irregular grid point is gradually extended by adding the base clouds from the valid neighbors. To incorporate the effect of the thin boundary layers, a wall modeling approach was chosen. This wall modeling approach was first successfully used to simulate the flow around the partially dressed landing gear for the AIAA Workshop on Benchmark problems for Airframe Noise Computations (BANC). A detailed discussion of this approach will be presented in Brehm et al.[14]. There are two additional challenges that had to be addressed in particular for moving geometries: (1) speeding up the geometry queries and regeneration of irregular stencils after moving the geometry and (2) the treatment of Freshly Cleared Cells (FCC). An x-ray algorithm, as visualized in Figure 1b, is used for inside-outside (in-out) testing. Parallel geometry kernels were implemented to allow fast in-out testing and exact computation of the distance from the irregular grid points to the surface triangulation (including point to plane and point to edge cases). An optimized Boundary Volume Hierarchy (BVH) based x-ray algorithm employing multi-resolution binning was developed for this purpose. Figure 1c displays some FCCs after moving the immersed boundary in the time interval $[t_n, t_{n+1}]$. Since the FCCs do not contain a valid time history, their state vector is initialized by interpolation considering the boundary condition and valid grid points (not including other FCC) in the neighborhood of the FCC. Some more advanced strategies for updating the FCCs similar to what has been discussed in Brehm and Fasel[15] for the incompressible Navier-Stokes equations are still being explored.

3   Outlook

First, the immersed boundary method by Brehm et al.[16, 17] is described and the necessary additions to efficiently capture unsteady body motion are introduced. Next, the structural Finite Element Solver (FEM) solver is introduced and its implementation is validated for simple static deformation and vibration problems. The validation section starts with moving boundary problems of prescribed motion. Finally, two examples of fully coupled fluid structure interaction problems are discussed. Experimental data is available for comparison for all moving boundary and fully coupled fluid structure interaction validation problems.

References

[1] C. Peskin. Numerical Analysis of Blood Flow in the Heart. *J. Comput. Phys.*, 25:220–252, 1977.

[2] C. S. Peskin. The Immersed Boundary Method. *Acta Numerica, Cambridge University Press*, pages 1–39, 2002.

[3] D. Goldstein, R. Handler, and L. Sirovich. Modeling a Non–Slip Flow Boundary with an External Force Field. *J. Comp. Phys.*, 105:354–366, 1993.

[4] Randall John LeVeque. *Time-Split Methods for Partial Differential Equations*. PhD thesis, Stanford University, 1982.

[5] A. Wiegmann and K. Bube. The Explicit–Jump Immersed Interface Method: Finite Difference Methods for PDEs with Piecewise Smooth Solutions. *SIAM J. Numer. Anal.*, 31(4):1019–1044, 1992.

[6] M. Linnick and H. Fasel. A High–Order Immersed Interface Method for Simulating Unsteady Incompressible Flows on Irregular Domains. *J. Comput. Phys*, 204:157–192, 2004.

[7] H. Johansen and P. Collela. A Cartesian Grid Embedded Boundary Method for Poisson's Equation on Irregular Domains. *Journal of Computational Physics*, 147:60–85, 1998.

[8] R. Mittal and G. Iaccarino. Immersed Boundary Methods. *Annu. Rev. Fluid Mech.*, 37:239–261, 2005.

[9] X. Zhong. A New High–Order Immersed Interface Method for Solving Elliptic Equations with Embedded Interface of Discontinuity. *Journal of Computational Physics*, 225:1066–1099, 2007.

[10] L. Duan, X. Wang, and X. Zhong. A High–Order Cut–Cell Method for Numerical Simulation of Hypersonic Boundary–Layer Instability with Surface Roughness. *Journal of Computational Physics*, 229:7207–7237, 2010.

[11] C. Kiris, M. Barad, J. Housman, E. Sozer, C. Brehm, and S. Moini-Yekta. The LAVA Computational Fluid Dynamics Solver. In *52st AIAA Aerospace Sciences Meeting, National Harbor, Maryland*, January 13-17, 2014. AIAA-2014-0070.

[12] M. Nemec and M. Aftosmis. Aerodynamic Shape Optimization Using a Cartesian Adjoint Method and CAD Geometry. *24th AIAA Applied Aerodynamics Conference, AIAA 2006-3456*, 2006.

[13] R. Mittal, H. Dong, M. Bozkurttas, F.M. Najjar, A. Vargas, and A. von Loebbecke. A Versatile Sharp Interface Immersed Boundary Method for Incompressible Flows with Complex Boundaries. *Journal of Computational Physics*, 227:4825–4852, 2008.

[14] C. Brehm, M.F. Barad, and C.C. Kiris. Towards a Viscous Wall Model for Immersed Boundary Methods. Submitted to AIAA Aviation and Aeronautics Forum and Exposition, Washington, DC, 2016.

[15] C. Brehm and H.F. Fasel. Immersed interface method for solving the incompressible navier stokes equations with moving boundaries. In *49th AIAA Aerospace Sciences Meeting, AIAA*, 2011. AIAA-2011-758.

[16] C Brehm, C Hader, and HF Fasel. A locally stabilized immersed boundary method for the compressible navier-stokes equations. *Journal of Computational Physics*, 295:475–504, 2015.

[17] C. Brehm and H.F. Fasel. A novel concept for the design of immersed interface methods. *Journal of Computational Physics*, 242(0):234 – 267, 2013.