Mitteilung

Fachgruppe: Numerische Aerodynamik

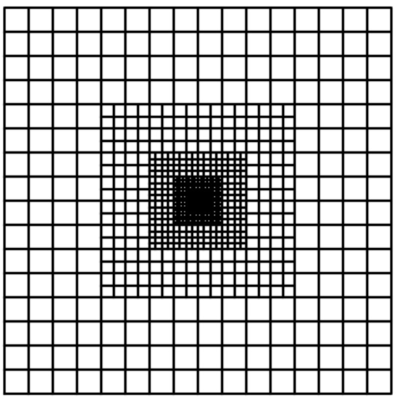
Towards „Rapid CFD“ via an Immersed Boundary Method in the CFD software by ONERA, DLR, Airbus (CODA)

Roberto Sanchez-Ortiz, Jesús Miguel Sánchez-Gil
DLR, Institut für Aerodynamik und Strömungstechnik
Bunsenstraße 10, 37073 Göttingen, Roberto.SanchezOrtiz@dlr.de
Lilienthalpl. 7, 38108 Braunschweig, Jesus.SanchezGil@dlr.de

In recent years, the industry has been looking for methods to simulate external aerodynamics at low overall turnaround times, such as Lattice-Boltzmann and Cartesian solvers. This paper proposes a “Rapid CFD” capability based on the Immersed Boundary Method (IBM) [1] in the unstructured CFD software by ONERA, DLR, Airbus (CODA). The main advantage of this method is a fully automatic mesh generation, which addresses this well-known bottleneck in industrial CFD applications.

The mesh generation consists of an octree algorithm where the mesh is automatically refined in the vicinity of the geometry, as shown in Figure 1. The generation is based on a near-wall size ($\Delta \mathrm{y}_{\mathrm{min}}$) and a growth parameter. The software used is Cassiopée [2], which generates a series of Cartesian blocks that are treated together as a single unstructured mesh with pseudo elements connecting the hanging faces between the different refinement levels. The boundary of this mesh does not conform to the geometry, leaving a gap between the computational boundary and the physical wall.

The IBM method in CODA follows the work of ONERA [3], [4], who impose the boundary condition directly without requiring any modification of the solver as those in [5], [6]. This is achieved by a hole-cutting algorithm [2] and a proper placement of the donor points [4]. Cells close to the wall are removed until the face centers of the immersed boundary, known as integration points, are immediately above a specified distance. This distance is specified by estimating a y^+ of 100, using the flat plate analogy. The donor points are located within the next cell. This location of the donor points provided the best results compared to other locations (such as constant distance to the wall). A full view of an octree mesh and a schematic view of the immersed points (donor, integration and wall) can be seen in Figure 1.



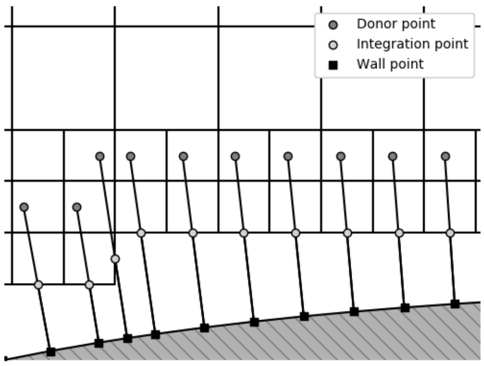


Figure 1. Octree mesh of NACA 0012 (left) and a close-up (right) showing donor, integration and wall points

The boundary conditions are imposed as follows, using the RANS equations and the negative Spalart-Allmaras (SAneg) turbulence model [7]. The fluid state at the donor point is recovered using a linear reconstruction in the cell in which the donor point is located. A wall function is then solved using the fluid state and distance of the donor point, and evaluated at the integration point, giving the velocity at the mesh boundary. This velocity is used as the parallel component with respect to the wall, while the normal component is interpolated linearly as the product of the normal component at the donor point and the ratio of distances from the wall of the integration point over the donor point. Density and pressure are assumed to be constant from the donor point, and the energy is computed accordingly using the equation of state for perfect gas. Finally, the SAneg variable is computed considering constant eddy viscosity as:

$$\tilde{\nu} = f_{\nu q}^{-1} \kappa y^{+}$$

The wall function used is the algebraic wall function of Musker [8], which includes the viscous sublayer, buffer, and logarithmic regions.

IBM was tested with a NACA 0012 airfoil subsonic case at a Mach number of 0.15 and Reynolds number of 6 million at zero angle of attack. The solution was compared against two references: a body-fitted (BF) simulation with fully resolved boundary layer ($y^{+} = 0.2$), and a coarser one using the Musker's wall function (BFWF) with a first cell height of $y^{+} = 100$. Figure 2 shows preliminary results of the present IBM, which gives good agreement with the fully resolved body-fitted case, in the leading-edge region even better than the body-fitted case with wall functions.

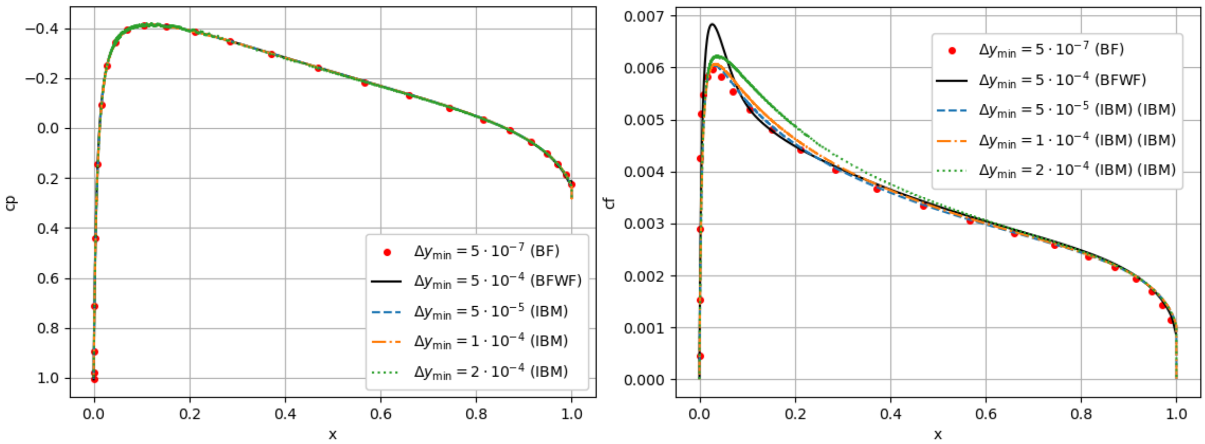


Figure 2. Pressure and friction distribution of NACA 0012, M=0.15, Re=6E6, AOA=0° for body-fitted (BF: circles), body-fitted with wall function (BFWF: solid), and IBM with three sizes (dashed, dashdot, and dotted).

Future applications will include 3D cases with complex geometries that pose significant challenges to mesh, as well as an extension to wall-modeled LES (WMLES) and Hybrid RANS-LES to enable scale-resolving simulations.

Acknowledgment CODA is the computational fluid dynamics (CFD) software being developed as part of a collaboration between the French Aerospace Lab ONERA, the German Aerospace Center (DLR), Airbus, and their European research partners. CODA is jointly owned by ONERA, DLR and Airbus.

Bibliography

[1] R. Mittal and G. Iaccarino, “Immersed Boundary Methods,” *Annu. Rev. Fluid Mech.*, vol. 37, no. 1, pp. 239–261, 2005, doi: 10.1146/annurev.fluid.37.061903.175743.

[2] C. Benoit, S. Péron, and S. Landier, “Cassiopee: A CFD pre- and post-processing tool,” *Aerosp. Sci. Technol.*, vol. 45, pp. 272–283, Sep. 2015, doi: 10.1016/j.ast.2015.05.023.

[3] S. Péron, C. Benoit, T. Renaud, and I. Mary, “An immersed boundary method on Cartesian adaptive grids for the simulation of compressible flows around arbitrary geometries,” *Eng. Comput.*, vol. 37, Jul. 2021, doi: 10.1007/s00366-020-00950-y.

[4] B. Constant, S. Péron, H. Beaugendre, and C. Benoit, “An improved immersed boundary method for turbulent flow simulations on Cartesian grids,” *J. Comput. Phys.*, vol. 435, p. 110240, Jun. 2021, doi: 10.1016/j.jcp.2021.110240.

[5] F. Capizzano, “Turbulent Wall Model for Immersed Boundary Methods,” *AIAA J.*, vol. 49, no. 11, pp. 2367–2381, Nov. 2011, doi: 10.2514/1.J050466.

[6] Y. Tamaki, M. Harada, and T. Imamura, “Near-Wall Modification of Spalart–Allmaras Turbulence Model for Immersed Boundary Method,” *AIAA J.*, vol. 55, no. 9, pp. 3027–3039, 2017, doi: 10.2514/1.J055824.

[7] S. Allmaras, F. Johnson, and P. Spalart, “Modifications and clarifications for the implementation of the Spalart-Allmaras turbulence model,” *Seventh Int. Conf. Comput. Fluid Dyn. ICCFD7*, pp. 1–11, Jan. 2012.

[8] A. J. Musker, “Explicit Expression for the Smooth Wall Velocity Distribution in a Turbulent Boundary Layer,” *AIAA J.*, vol. 17, no. 6, pp. 655–657, Jun. 1979, doi: 10.2514/3.61193.