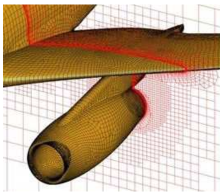
Numerical solution of the incompressible Navier-Stokes equations with the immersed-boundary technique

Giuseppe Bonfigli and Patrick Jenny, Institute of fluid Dynamics, ETH-Zurich, CH-8092 Zurich
bonfigli@ifd.mavt.ethz.ch   jenny@ifd.mavt.ethz.ch

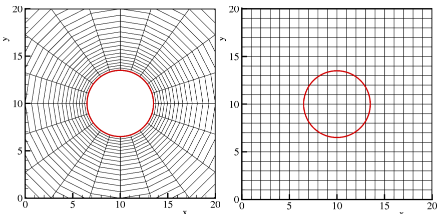
Basic idea

The generation of body fitted grids for finite-volume computations is typically the most time-consuming task in the work-flow of industrial CFD applications. Frequently involved geometries are so complex that high quality grids are not affordable and computations are carried out considering grids with degenerated cells, thus affecting the quality of the results and undermining the stability of the numerical procedure.



Complex unstructured grid around an airplane wing.

The immersed-boundary approach removes these difficulties by considering non-fitted orthogonal meshes. In particular, we consider a finite-difference discretization, and impose no-slip boundary conditions at solid walls exactly by modifying difference stencils intersecting the boundary. This is achieved in a fully automatic manner. The only geometrical information required is the distance between grid nodes and intersection points of the grid lines with the boundary. Normal direction, curvature, and higher differential properties of the boundary surface need not be known to achieve high order accuracy.



Body-fitted (left) and immersed grid (right).

Governing equations

The numerical procedure is based on a primitive-variable formulation of the incompressible Navier-Stokes equations on staggered grids (Harlow & Welch, 1965). Second or fourth order finite-differences are considered for spatial discretization. The third-order Adams-Bashforth scheme or the standard fourth-order Runge-Kutta scheme are used for time integration.

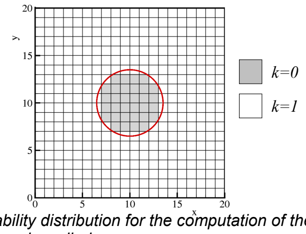
Dirichlet boundary conditions are imposed on all velocity components and the pressure is computed from a Poisson equation equivalent to the continuity equation:

$$\nabla\cdot(k\nabla p)=R(\underline{u}). \qquad (1)$$

Boundary conditions on p are indirectly derived from the constraints on the velocity, which are taken into account when computing the right hand side of (1).

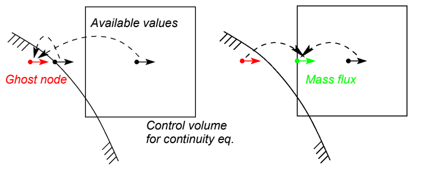
Solution of the pressure equation

Equation (1) is solved iteratively combining a fix-point iteration with the MSFV-procedure (insert) used as pre-conditioner. Immersed solid bodies are represented by zero-permeability regions ($k=0$).



Permeability distribution for the computation of the flow around a cylinder.

Boundary conditions on the velocity are considered by means of ghost velocity nodes, where velocity components are estimated by extrapolation, considering prescribed values at the boundary and values from neighbouring nodes lying inside the flow region.

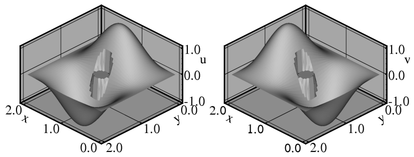


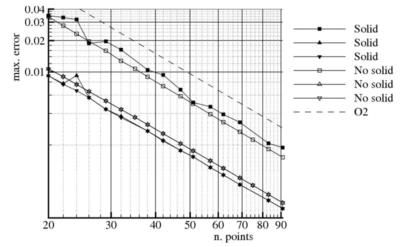
Interpolation to ghost node (left) and flux evaluation at cell boundaries as considered in the MSFV-procedure (right).

Values at ghost nodes depend on the unknown solution for the pressure and have to be recomputed at every iteration step.

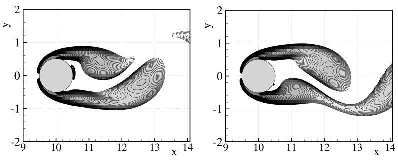
Validation of the procedure

Results are presented for two test cases. In the first set of computations a steady solution is prescribed for Re=1 by introducing an appropriate volume force. Second order convergence with respect to the spatial discretization is verified by comparing numerical and exact solutions. In the second test case the flow around a cylinder at Re=100 is computed. An unsteady self-sustained Karman-street develops as expected after initial triggering.





Solution (top) and convergence behaviour with respect to spatial discretization (bottom) for the volume-force case.



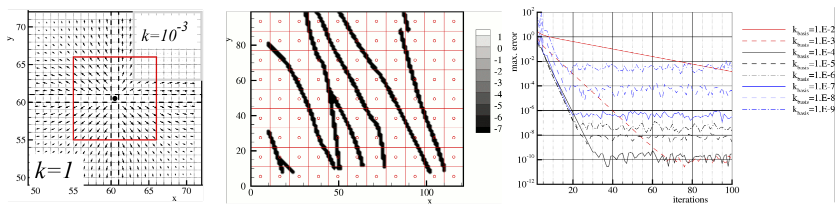
Vorticity iso-coturs for the flow around a cylinder at Re=100 (snap-shot at two different time steps).

The MSFV-technique

The multi-scale-finite-volume technique (MSFV, Jenny et al., 2003) has been developed to solve large linear systems resulting from the finite-volume discretization of the elliptic problems with strongly non-homogeneous coefficients:

$$\nabla\cdot(k\nabla p) \qquad\qquad k=k(x,y).$$

The procedure has some analogies with the multi-grid approach, but prolongation and restriction are based on numerically computed basis functions. This allows for large upscaling factors (5 to 20 fine cells per coarse cell) keeping account of spatial changes in the permeability *k with sufficient accuracy also on the coarse grid*.



Fluxes induced by a basis function in the case of discontinuous permeability (left). Permeability distribution for a test case (center) and corresponding convergence rates for the MSFV procedure considering different definitions of the basis functions (right).

The coarse grid solution is determined by imposing flux balance over a set of properly chosen control volumes (coarse cells, red lines in the figures). This ensures global coupling over the whole integration domain. On the other hand, fine-scale convergence is enforced by localized relaxation on the fine grid (line relaxation or domain decomposition).