An optimal, fast, parallel solver for the incompressible Navier–Stokes equations on colocated grids

Franco Auteri [*1], Alessandro Chiarini[1], and Maurizio Quadrio[1]
[1]*Department of Aerospace Sciences and Technologies, Politecnico di Milano, via La Masa 34, 20156 Milano, Italy*

Summary A novel colocated finite difference solver for the incompressible Navier-Stokes equations is presented, based on the direction-splitting fractional step method proposed in [Guermond & Minev 2010]. High computational efficiency is obtained using the direction-splitting method; high scalability is obtained thanks to the Schur complement method used for parallelisation. The checkerboard pressure modes have been stabilised using the approach introduced by [Rhie & Chow 1983], which consists in adding a fourth order term to the right-hand side of the pressure equation and using a short stencil for second derivatives in the left-hand side. The obtained solver is found to be second order accurate in time and space. As expected, the new solver is faster than the staggered version and its parallel performance scales extremely well with the number of processors.

The large increase in CPU and memory performance of recent computers leads to new opportunities for Direct Numerical Simulations (DNS) of turbulent flows. An example is the DNS of complex biomedical flows such as the flow in the nasal cavities [2] or the blood stream [3]. These new possible applications of the DNS require large grids – up to 10^9 points – and high geometrical flexibility. Therefore, together with improving computer performance, suitable new DNS programs need to be developed capable of taking advantage of the hardware performance and allowing an increased geometrical flexibility. In this respect, the time and space discretisation methods can be key to fully exploit the available performance to solve problems still elusive. In particular, when fluid structure interaction problems within complex geometries are concerned, such as in biomedical applications, not only high geometrical flexibility is needed, but also the capability to cope with a shape that changes with the flow. Finite elements, despite providing high geometrical flexibility and good accuracy, lack the necessary efficiency when the mesh must be modified in each time step. Finite differences, able to deal with such complex geometries by the immersed boundary method, seem to provide the best trade-off between accuracy and efficiency in these applications.

A very efficient finite difference solver for the incompressible Navier–Stokes equations on staggered grids has been recently introduced by Guermond and Minev [1], the solver has an optimal computational complexity and is highly scalable. The efficiency of this solver hinges on a novel fractional-step method used to decouple the viscous step from the incompressibility constraint that allows the use of the ADI algorithm [4] for both the viscous and the pressure step. Using a Crank–Nicolson time discretisation, the viscous step reads:

$$\frac{\boldsymbol{\xi}^{n+1}-\boldsymbol{u}^n}{\Delta t}-\frac{1}{\mathrm{Re}}\nabla^2\tilde{\boldsymbol{u}}^n=f^{n+1/2}-\nabla p^{*,n+1/2}-\mathbf{nl}(\boldsymbol{u}^{n+1/2}) \tag{1}$$

$$\frac{\boldsymbol{\eta}^{n+1}-\boldsymbol{\xi}^{n+1}}{\Delta t}-\frac{1}{2\mathrm{Re}}\frac{\partial^2}{\partial x^2}\left(\boldsymbol{\eta}^{n+1}-\boldsymbol{u}^n\right)=0 \tag{2}$$

$$\frac{\boldsymbol{\zeta}^{n+1}-\boldsymbol{\eta}^{n+1}}{\Delta t}-\frac{1}{2\mathrm{Re}}\frac{\partial^2}{\partial y^2}\left(\boldsymbol{\zeta}^{n+1}-\boldsymbol{u}^n\right)=0 \tag{3}$$

$$\frac{\boldsymbol{u}^{n+1}-\boldsymbol{\zeta}^{n+1}}{\Delta t}-\frac{1}{2\mathrm{Re}}\frac{\partial^2}{\partial z^2}\left(\boldsymbol{u}^{n+1}-\boldsymbol{u}^n\right)=0 \tag{4}$$

where $\boldsymbol{u}^n$ is the velocity at time t_n, $p^{*,n+1/2}=p^{n-1/2}+\phi^{n+1/2}=2p^{n-1/2}-p^{n-3/2}$ is the pressure prediction at time $t_{n+1/2}$, Re is the Reynolds number and **nl** the nonlinear term. Instead, the pressure step reads:

$$\psi-\frac{\partial^2\psi}{\partial x^2}=-\frac{1}{\Delta t}\nabla\cdot\boldsymbol{u}^{n+1}, \qquad \frac{\partial\psi}{\partial x}|_{x=0,1}=0, \tag{5}$$

$$\varphi-\frac{\partial^2\varphi}{\partial y^2}=\psi, \qquad \frac{\partial\varphi}{\partial y}|_{y=0,1}=0, \tag{6}$$

$$\phi^{n+1/2}-\frac{\partial^2\phi^{n+1/2}}{\partial z^2}=\varphi, \qquad \frac{\partial\phi^{n+1/2}}{\partial z}|_{y=0,1}=0. \tag{7}$$

This method retains the efficiency of $1D$ solvers with banded matrices, hence an optimal computational complexity, in a direct solver. The high scalability is guaranteed by the use of the Schur-complement method [1].

While the capabilities of this solver have been extended to complex geometries by a Moving Least Square immersed boundary procedure [6], the use of a staggered grid to enforce the LBB condition prevents the potential of the solver to be fully exploited when moving geometries and fluid-structure interaction problems are concerned. Indeed, when applying the immersed boundary procedure, the calculation of all the coefficients necessary for the involved interpolations must be computed on the four staggered grids, leading to a significant performance degradation.

[*]Corresponding author. E-mail: franco.auteri@polimi.it.

These problems motivated us to develop a colocated version of the aforementioned fractional-step solver. The instability related to pressure spurious modes has been cured by the method proposed by Faure et al. [8] which heavily relies on the well known method first proposed by Rhie and Chow [7]. It consists in adding a stabilisation term in the form of a fourth derivative term multiplied by an under-relaxation coefficient in the right-hand side of the equivalent of the Poisson equation for pressure. The stabilisation term reads:

$$\alpha c(\nabla^2_L p - \nabla^2_s p) \tag{8}$$

where $\nabla^2_s p$ is the pressure Laplacian computed on the small stencil – in 2D $(p_W - 2p_C + p_E)/\Delta x^2 + (p_S - 2p_C + p_N)/\Delta y^2$, see fig 1 – and $\nabla^2_L p$ is the pressure Laplacian computed on the large stencil – $(p_{WW} - 2p_C + p_{EE})/4\Delta x^2 + (p_{SS} - 2p_C + p_{NN})/4\Delta y^2$– and $\alpha \in [0, 1]$ and c a constant.

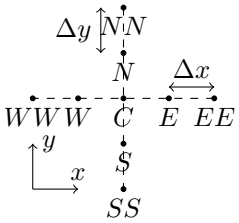

Figure 1: Colocated grid in the 2D case.

The stabilisation term partially substitutes the large-stencil Laplacian of the pressure implicitly present in the right-hand side of the pressure equation with its small-stencil counterpart, thus coupling all grid points.

As expected, the colocated program is slightly less accurate than the staggered one, but second order convergence is obtained in both time and space for velocity and pressure (see figure 2). Moreover, the colocated program displays faster

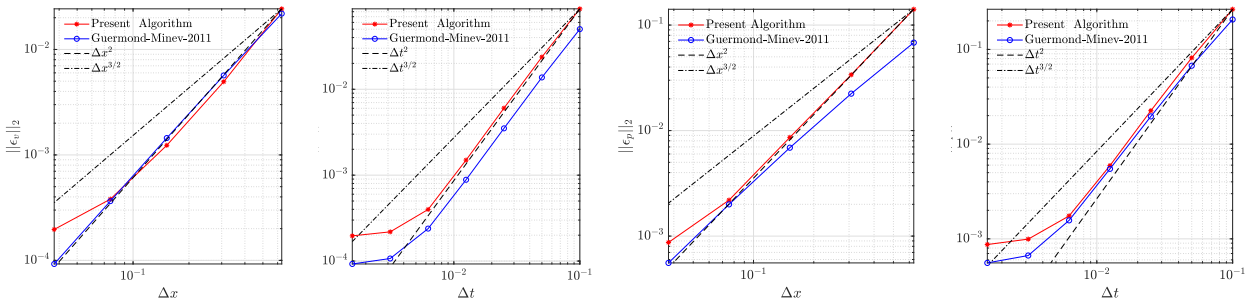

Figure 2: Convergence curves. Top-left panel: spatial convergence of the velocity; top-right panel: time convergence of the velocity; bottom-left panel: time convergence of the velocity; bottom-right: time convergence of the pressure

convergence in space with respect to the original staggered one (see bottom-left in figure 2).

In addition, the colocated version of the program is about 30% faster than the staggered one, and still presents high scalability by virtue of the Schur complement method used for parallelisation.

References

[1] Guermond J.L., Minev P.D. A new class of fractional step techniques for the incompressible Navier–Stokes equations using direction splitting. *Compt. Rendus Math,* **348**: 581-585, 2010.

[2] Covello V., Pipolo C., Saibene A., Felisati G., Quadrio M. Numerical simulation of thermal water delivery in the human nasal cavity. *Comput. Biol. Med.* **100**: 62-73, 2018.

[3] Crosetto P., Deparis S., Fourestey G., Quarteroni A. Parallel algorithms for fluid-structure interaction problems in haemodynamics. *SIAM J. Sci. Comput.* **33**: 1598-1622, 2011.

[4] Douglas J. Alternating direction methods for three space variables. *Numer. Math.* **4**, 41-63, 1962.

[5] Guermond J.L., Minev P.D. A new class of massively parallel direction splitting for the incompressible Navier–Stokes equations. *Comput. Methods Appl. Mech. Engrg.* **2000**: 2083-2093, 2011.

[6] Auteri F., De Tullio M.D., Guermond J.-L., Montagnani D., Konghar P.D. A Massively Parallel, Direction-Splitting Solver for DNS in Complex Geometries. Salvetti M.V., Armenio V., Fröhlich J., Geurts B.J., Kuerten H. eds., *Direct and Large-Eddy Simulation XI*, Springer, 23-29, 2019.

[7] Rhie C.M.,Chow W.L. Numerical Study of the Turbulent Flow past an Airfoil with Trailing Edge Separation. *AIAA J.* **21**: 1525-1532, 1983.

[8] Faure S., Laminie J., Temam R. Colocated finite volume schemes for fluid flows. *Comm. Comput. Phys.* **4**: 1-25, 2008.