

An implicit multiblock coupling for the incompressible Navier–Stokes equations

Christophe Romé^{*,†} and Stéphane Glockner[‡]

*Laboratoire Transferts Écoulements Fluides Énergétique (TREFLE) UMR CNRS 8508, Site ENSCPB,
16 Avenue Pey-Berland, 33607 Pessac Cedex, France*

SUMMARY

In the present work, we describe an interface treatment of multiblock computation for the incompressible Navier–Stokes equations discretized upon a finite volume marker and cell (MAC) approach. The connection between the different blocks is based on velocity interpolation only. The originality of this work is that the polynomial interpolation coefficients are present in the final linear system and ensure that the equations are globally resolved. Copyright © 2005 John Wiley & Sons, Ltd.

KEY WORDS: multiblock; block-structured; matching, non-matching and overlapping grids; polynomial interpolation; Navier–Stokes

1. INTRODUCTION

Numerical difficulties are often encountered when treating complex geometries that are common in many industrial processes such as turbomachinery flows or moulding simulations.

A common practice to treat such complex geometries is based on the adoption of a monoblock grid and penalization of Navier–Stokes equations using for example the Brinkman term $-(\mu/K)\mathbf{V}$ [1]. K is the permeability of the continuum that varies from zero (solid) to infinity (fluid). However, this technique can be used to describe simple geometries (Figure 1(a)), but proves insufficient for complex ones and can lead to an increase in computation time due to additional nodes. Moreover, for complex geometries, one-block meshes often lead to poor mesh quality and insufficient resolution. Multiblock techniques make up for these three drawbacks, owing to the division of the topology into subdomains that are meshed independently. Each block is connected to its adjacent ones so that the entire domain can be solved.

*Correspondence to: Christophe Romé, Laboratoire TREFLE UMR CNRS 8508, Site ENSCPB, 16, Avenue Pey-Berland, 33607 Pessac Cedex, France.

[†]E-mail: rome@enscpb.fr

[‡]E-mail: glockner@enscpb.fr

Received 27 April 2004

Revised 27 October 2004

Accepted 29 November 2004

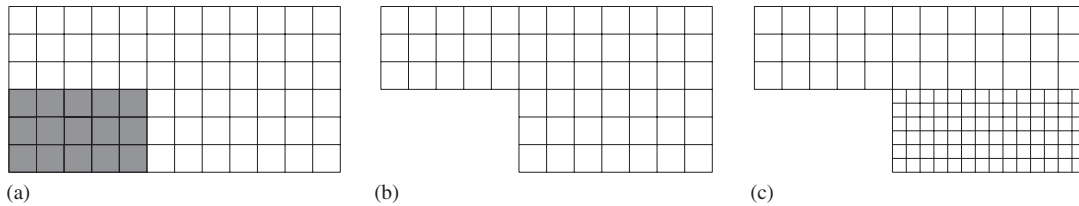


Figure 1. Example of monoblock and multiblock grids: (a) monoblock grid; (b) multiblock grid (matching blocks); and (c) multiblock grid (non-matching blocks).

A multiblock grid can be made up of matching and non-matching blocks. In the first case, whatever the mesh line, it is continuous throughout the interfaces (see Figure 1(b)). In the second case, as one follows a line, it can stop at the interface due to a discontinuity of the mesh (see Figure 1(c)). It is interesting to observe that non-matching meshes are locally structured but globally unstructured. In both cases, blocks can either overlap or not. The interface of non-overlapping blocks is a line (or a surface in 3D), and a surface (or a volume in 3D) for the overlapping case.

An interesting state of art on the multiblock theme is presented in Reference [2]. Most authors use finite element or finite difference methods with an explicit resolution and use an iterative process to solve the interface treatment, each block being computed separately. Many related works refer to several factors that have an influence on the accuracy of the solution, such as the order of polynomial interpolation, the grid resolution, and the conservative or non-conservative behaviours [3–5]. Other authors [2, 6] work on finite volume discretizations, using a pressure-based method to compute Navier–Stokes and have studied the coupling of the pressure between blocks.

The goal of the present work is to propose an easy and original technique of coupling between blocks. This will be done within the scope of the CFD code Aquilon [7–12], based on a MAC orthogonal curvilinear structured grid. An implicit finite volume method is used, with a centred scheme for the spatial discretization. In order to deal with the pressure/velocity coupling and the divergence-free velocity, we use the augmented Lagrangian method [13]. The incompressibility constraint has been introduced in the momentum equations thanks to the term $-r\nabla(\nabla \cdot u)$, the pressure components being considered as Lagrange multipliers. An iterative Uzawa algorithm solves the augmented Lagrangian problem and gives the pressure explicitly by

$$p^{n+1} = p^n - r\nabla \cdot u^{n+1} \quad (1)$$

So, for the multiblock treatment, nothing must be done on pressure.

2. PROPOSED METHOD

If we consider first the multiblock matching problem, the idea of the present method is to delete for each twin node one of them, and then to reconstruct the incomplete stencil of the remaining nodes (see Figure 2(a) for 1D problem). Thanks to the indirect addressing,

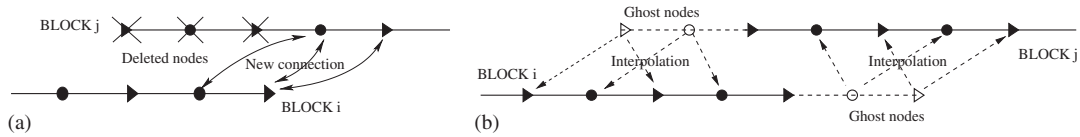


Figure 2. 1D treatment of connection between two blocks: (a) matching blocks; and (b) non-matching blocks.

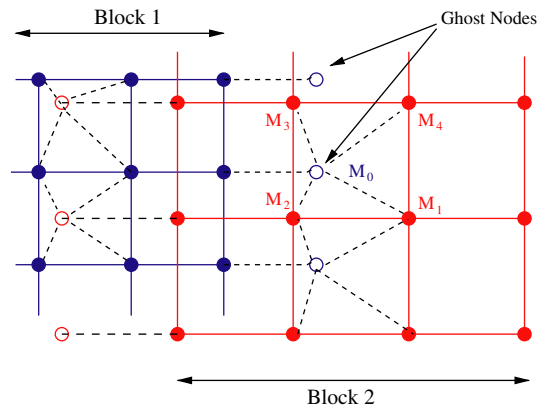


Figure 3. Representation of the 'spider' mesh.

this method is straightforward and the results should be identical to those obtained with a monoblock grid with obstacle penalization.

Tests have been performed on a series of academic cases such as Poiseuille flow, backward-facing step flow, driven cavities, etc. Industrial simulations have also been carried out and results have been found to be identical to the monoblock ones. For example, in the case of propellant filling of solid propulsion boosters of Ariane 5 [9], the 3D grid includes 216 000 nodes if obstacles are meshed, and 180 668 if not. This corresponds to a saving of about 20 per cent in calculation time and memory.

For the multiblock non-matching grids, using the nodes of the adjacent blocks to complete the stencil would result in a deformation of the cell and would not maintain the conservation of the fluxes. In the present approach, for each cell at the interface between two blocks, a new one is created, called a ghost cell (see Figure 2(b)). These nodes are not real nodes of the discretization: the velocity will be calculated there by an implicit polynomial interpolation using the values of the adjacent blocks. These ghost cells join a block to the adjacent one by a kind of 'spider' mesh as shown in Figure 3 for a 2D representation.

As the computation of the junction of the blocks is implicit, the interpolation is present in the linear system and modifies the structure of the matrix. In 2D, the original matrix is made up of two blocks (one for each component of the vector velocity), coupled by extra diagonals coming from an augmented Lagrangian method. New unknown variables, \tilde{u} and \tilde{v} ,

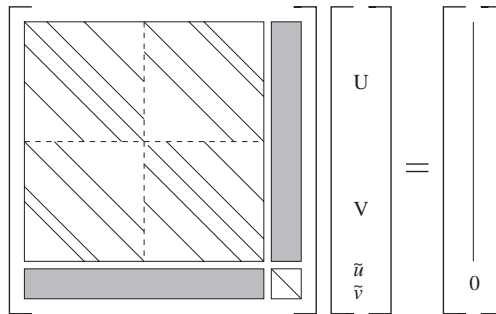


Figure 4. Schematic representation of the Navier–Stokes matrix with the additional blocks resulting from the implicit interpolation.

corresponding to the ghost cells are included at the end of the matrix as a new horizontal block that contains the interpolation polynomials (see Figure 4).

Polynomial interpolation is a widely known method to determine the value of a field $\phi(x, y)$ at a point $M_0(x_0, y_0)$, whatever its position on the grid (see Figure 3). The technique consists of the construction of a subcanonical basis of Q-type polynomials of order k ($k = 1, 2, 3$ in the present study) thanks to the surrounding nodes of the point $M_0(x_0, y_0)$. The number of nodes required depends on the order of the polynomial (for instance, Q2 requires 27 nodes in 3D, and Q3, 64).

A polynomial Q_i^k is associated with each necessary surrounding node $M_i, 1 \leq i \leq (k + 1)^2$ in 2D:

$$Q_i^k(x - x_0, y - y_0) = \sum_{m=0}^k \sum_{n=0}^k a_{m,n,i} (x - x_0)^m (y - y_0)^n \quad (2)$$

that has the following properties:

$$\forall i, j, 1 \leq i, j \leq (k + 1)^2; \quad Q_i^k(x_j - x_0, y_j - y_0) = \delta_{ij} \quad (3)$$

A $(k + 1)^2 \times (k + 1)^2$ linear system is built, Equation (3) being a line of the matrix. Its inversion gives coefficients $a_{m,n,i}$. A notable property is that the summation on m and n of the coefficients $a_{m,n,i}$ is equal to one. The value of the field ϕ at point $M(x_0, y_0)$ is given by

$$\phi(x_0, y_0) = \sum_{i=0}^{(k+1)^2} Q_i^k(0, 0) \phi(x_i, y_i) \quad (4)$$

The inversion of the local linear system, for each ghost node, is performed once, during the preparation of the computation, before the resolution time loop. The values $Q_i^k(0, 0)$ of the different polynomials go directly into the global linear system of the velocity vector (see Figure 4).

3. RESULTS AND PROSPECTS FOR NON-MATCHING GRIDS

First, we have analysed different polynomial interpolation behaviours using several analytical solutions: Couette and Poiseuille flows, and a sheared vortex field.

The independence of the interpolation precision on the sharpness of the interpolation grid has been verified by interpolating the analytical solutions from a 16×16 grid onto 32×32 , 64×64 and 128×128 grids. As expected, for a solution with the same order (or lower) than the order of the interpolation polynomial, the observed error has been found around the computer double precision. That means also that the Q1 polynomial should not be used because of its incapacity to solve the Poiseuille flow (error of 9.45D-4).

The opposite exercise has been then performed, that is to say, the passage from the analytical solution on five grids (from 20×20 to 100×100), to a unique interpolation grid of 128×128 . For the vortex solution, that is the most interesting, the convergence order, on the velocity field and the divergence error (see Table I), conforms to the theory (super convergence). It can be noted that the divergence error decreases with the increase in the polynomial order. For a code that is globally at a space precision of order two, the only interest to use Q3 interpolation would be to obtain an improvement of the divergence. Even though we have no difficulties in the inversion of the Navier-Stokes linear system due to the added lines, it is important to notice that to compute a component of the velocity vector at any point of the domain, all components of the surrounding nodes are interpolated. So, a line of the matrix can have up to 196 elements (in 3D with a Q3 interpolation).

Poiseuille flow simulations using several multiblock grids represented in Figure 5 have been performed. These five grids ensure that all types of non-conforming blocks have been tested, from the case where half of the points conform (cases 1 and 2), to full non-conforming blocks (case 5).

It has been observed that stationarity and residual parameters have reached computer precision. Good results have been obtained for the maximal divergence and the maximal

Table I. Interpolation error and convergence order on velocity and divergence for the sheared vortex solution.

	Error on \mathbf{u}	Error on $\text{div}(\mathbf{u})$	Convergence order for \mathbf{u}	Convergence order for $\text{div}(\mathbf{u})$
Q1	1.61E-06	2.03E-05	2.004	1.773
Q2	2.30E-08	2.88E-09	3.006	2.840
Q3	3.35E-12	4.62E-11	4.015	3.830

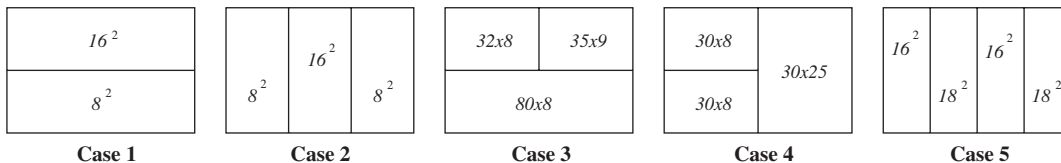


Figure 5. A schematic representation of the configuration of the blocks; the number of nodes by block is indicated.

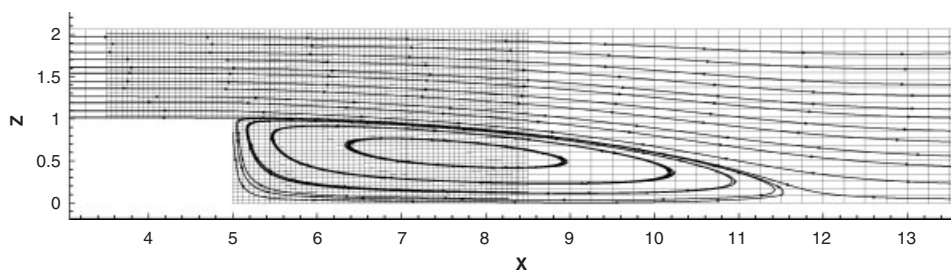


Figure 6. Streamlines of the backward-facing step flow at $Re = 300$.

error on the velocity field (from 3.7D-12 to 3.9D-14), whatever the grid and polynomial choices.

The second test chosen is the laminar backward-facing step flow. The domain has been divided into three blocks, the middle one being three times finer than the others. Figure 6 shows the main recirculation and the continuity of the flow between the blocks. The length of the recirculation at different Reynolds numbers ($Re = 100, 200$ and 300) is 2.92, 4.88 and 6.70, in agreement with the experimental results reported by Armaly [14] (2.86, 4.86 and 6.57). We have tested different time and spatial steps, viscosities, densities, etc. and we have not found any stability problem.

4. CONCLUSION AND PERSPECTIVES

An implicit non-matching multiblock coupling has been proposed for the solution of the incompressible Navier–Stokes equations. Ghost cells are added at the interface between two blocks. The velocity values at these nodes are calculated using a Q2 or Q3 polynomial interpolation. The values of the polynomials at the ghost nodes are present in the final linear system and assure the coupling between the adjacent blocks. First results are encouraging and show the feasibility of this approach. Further research work should consider the effect of the choice between Q2 and Q3 polynomials, on both velocity and divergence. It could also be interesting to test P2 and P3 polynomials that have less cross terms. It has been observed in the present study that divergence in the whole domain is controlled by divergence at the interface. It should be interesting to improve the null divergence property of the interpolation. Finally, the present method can be applied to other meshing techniques such as multigrid or AMR which share part of their problematics with multiblock ones.

REFERENCES

1. Arquis E, Caltagirone J-P. Sur les conditions hydrodynamiques au voisinage d'une interface milieu fluide—milieu poreux: application à la convection naturelle. *Comptes Rendus de l'Académie des Sciences Paris* 1984; **299**(série II):1–4.
2. Liu J, Shyy W. Assessment of grid interface treatments for multi-block incompressible viscous flow computation. *Computers and Fluids* 1996; **25**(8):719–740.
3. Cheshire G, Henshaw WD. A scheme for conservative interpolation on overlapping grids. *SIAM Journal on Scientific Computing* 1994; **15**(4):819–845.

4. Henshaw WD. A fourth-order accurate method for the incompressible Navier-Stokes equations on overlapping grids. *Journal of Computational Physics* 1994; **113**:13–25.
5. Durst F, Schfer M. A parallel block-structured multigrid method for the prediction of incompressible flows. *International Journal for Numerical Methods in Fluids* 1996; **22**:549–565.
6. Coelho PJ, Duic N, Lemos C, Carvalho MG. Modelling of a solid fuel combustion chamber of a ramjet using a multi-block domain decomposition technique. *Aerospace Science and Technology* 1998; **2**:107–119.
7. Vincent S, Caltagirone J-P. Efficient solving method for unsteady incompressible interfacial flow problems. *International Journal for Numerical Methods in Fluids* 1999; **30**(6):795–811.
8. Vincent S, Caltagirone J-P, Lubin P, Randrianarivelo N. An adaptative augmented Lagrangian method for three dimensional multi-material flows. *Computers and Fluids* 2004; **33**:1273–1289.
9. Breil J. Modélisation du remplissage en propergol de moteur à propulsion solide. *thèse de doctorat*, Université de Bordeaux 1, 2001.
10. Vincent S, Caltagirone J-P. Sur une méthode de pénalisation tensorielle pour la résolution des équations de Navier-Stokes. *Comptes Rendus de l'Académie des Sciences Paris* 2001; **329**(série IIb):607–613.
11. Caltagirone J-P, Breil J. Sur une méthode de projection vectorielle pour la résolution des équations de Navier-Stokes. *Comptes Rendus de l'Académie des Sciences Paris* 1999; **327**(série IIb):1179–1184.
12. Khadra K, Parneix S, Angot P, Caltagirone J-P. Fictitious domain approach for numerical modelling of Navier-Stokes equations. *International Journal for Numerical Methods in Fluids* 2000; **34**:651–684.
13. Fortin M, Glowinski R. Méthodes de lagrangien augmenté. Applications à la résolution de problèmes aux limites. In *Collection Méthodes Mathématiques de l'Informatique*. Ed. Dunot, 1982.
14. Armaly P-H, Durst F, Pereira J-CF, Schonung F. Experimental and theoretical investigation of backward-facing step flow. *Journal of Fluid Mechanics* 1983; **127**:473–496.