Journal of Computational and Applied Mathematics 235 (2010) 460–469

Contents lists available at ScienceDirect

Journal of Computational and Applied Mathematics

journal homepage: www.elsevier.com/locate/cam

Time stepping for vectorial operator splitting

Rossitza S. Marinova [a,*], Raymond J. Spiteri [b], Eddy Essien [a]

[a] Department of Mathematical and Computing Sciences, Concordia University College of Alberta, 7128 Ada Boulevard, Edmonton, AB, T5B 4E4, Canada

[b] Department of Computer Science, University of Saskatchewan, 176 Thorvaldson Bldg, 110 Science Place, Saskatoon, SK S7N 5C9, Canada

ARTICLE INFO

Article history:
Received 30 October 2009
Received in revised form 1 April 2010

MSC:
76D05
34A09
65M12

Keywords:
Unsteady incompressible Navier–Stokes
Implicit method
Stability

ABSTRACT

We present a fully implicit finite difference method for the unsteady incompressible Navier–Stokes equations. It is based on the one-step θ-method for discretization in time and a special coordinate splitting (called vectorial operator splitting) for efficiently solving the nonlinear stationary problems for the solution at each new time level. The resulting system is solved in a fully coupled approach that does not require a boundary condition for the pressure. A staggered arrangement of velocity and pressure on a structured Cartesian grid combined with the fully implicit treatment of the boundary conditions helps us to preserve the properties of the differential operators and thus leads to excellent stability of the overall algorithm. The convergence properties of the method are confirmed via numerical experiments.

1. Introduction

Fluid flows with high Reynolds numbers or complex geometries are challenging to simulate and of great interest to industry; hence there is significant demand for robust and stable algorithms and software, perhaps even at the expense of a moderately increased computational cost. Fully implicit time-stepping methods are generally more robust and stable than the explicit and semi-explicit methods. Therefore, as suggested in [1], fully implicit methods should be further investigated and developed.

The most popular time-stepping methods for the Navier–Stokes equations are the so-called *projection* or *operator splitting* methods (e.g., fractional step or pressure-correction methods) and are not fully implicit; see [2,1]. Decoupling the velocity and pressure reduces the system into simpler sub-problems, but the choice of boundary conditions for the pressure in these procedures is problematic. Moreover, the explicit element introduced by this decoupling requires small time steps to maintain stability. Although operator splitting methods can work well, they must be used with care in terms of how well the overall solution algorithm behaves. They are usually not suitable for flows with high Reynolds numbers or long simulation times because the requirement of a small time step size.

After discretization in space and time, a fully implicit approach leads to a system of nonlinear equations that may be singular [1]. For this reason, special spatial discretization or stabilization techniques are needed. Strongly coupled solution strategies can improve the stability considerably; however, they also need to be able to handle large nonlinear algebraic systems. Direct solvers can be used for the solution of the linear systems of equations that arise in this process, but they typically require large amounts of memory, and despite increases in computational power, are still not feasible for large-scale computations, particularly for unsteady 3D problems. Hence iterative solvers are the preferred choice for the solution of these systems. Coordinate splitting and multigrid are two powerful methods for solving such systems.

* Corresponding author. Tel.: +1 780 378 8430; fax: +1 780 474 1933.
E-mail addresses: rmarinova@math.concordia.ab.ca (R.S. Marinova), spiteri@cs.usask.ca (R.J. Spiteri), eessien@csa.concordia.ab.ca (E. Essien).
URL: http://www.math.concordia.ab.ca/marinova/ (R.S. Marinova).

0377-0427/$ – see front matter © 2010 Elsevier B.V. All rights reserved.
doi:10.1016/j.cam.2010.05.049

In this paper, we use the linear two-layer (one-step) scheme, which is also known as the θ-method, for the temporal discretization; see e.g., [3]. We employ finite difference approximations in space that utilize computer resources effectively and hence enable efficient computations. For the solution of the nonlinear stationary problems that arise after the temporal discretization, we use coordinate splitting based on the Douglas–Rachford scheme [4]. The splitting procedure is constructed in a way that leaves the system coupled to allow the satisfaction of the boundary conditions but avoids the introduction of artificial boundary conditions for the pressure.

The paper is organized as follows. The problem is formulated in the next section. The time discretization is presented in Section 3, including a discussion on the singularity of direct fully implicit schemes. Issues associated with the solution of the stationary problems that need to be solved after discretization in time are discussed in Section 4. These include requirements to be satisfied by the differential problem and the choice of discretization in space as well as the coordinate splitting method. Finally, numerical results are presented in Section 5 and conclusions in Section 6.

2. Problem statement

2.1. Incompressible Navier–Stokes equations

We consider the multi-dimensional incompressible Navier–Stokes equations in dimensionless form

$$\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = \nu \nabla^2 \boldsymbol{u} - \nabla p + \boldsymbol{g} \tag{1}$$

coupled with the continuity equation, also called the incompressibility condition,

$$\operatorname{div} \boldsymbol{u} = \nabla \cdot \boldsymbol{u} = 0 \tag{2}$$

on $\Omega \times (0, T)$, where Ω is a bounded, compact (spatial) domain with a piecewise smooth boundary $\partial\Omega$. Here $\boldsymbol{u} = \boldsymbol{u}(\boldsymbol{x}, t) = (u, v, w)$ is the fluid velocity at position $\boldsymbol{x} \in \Omega$ and time $t \in (0, T)$ for given T. Also $p = p(\boldsymbol{x}, t)$ is the fluid kinematic pressure, $\nu = 1/\mathrm{Re}$ is the kinematic viscosity, where Re is the Reynolds number, $\boldsymbol{g}$ is an external force, ∇ is the gradient operator, and ∇^2 is the Laplacian operator.

We can write the momentum equation (1) in the following form,

$$\frac{\partial \boldsymbol{u}}{\partial t} + (C + L)\boldsymbol{u} + \nabla p = \boldsymbol{g}, \tag{3}$$

where $C = \boldsymbol{u} \cdot \nabla$ is the nonlinear convection operator and $L = -\nu\nabla^2$ is the linear viscosity operator.

Taking into account the incompressibility constraint (2), the nonlinear convective term $(\boldsymbol{u} \cdot \nabla)\boldsymbol{u}$ in Eq. (1) can be written in the equivalent form

$$\begin{aligned} C\boldsymbol{u} &= (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} + \frac{1}{2}\boldsymbol{u}(\nabla \cdot \boldsymbol{u}) \\ &= \nabla \cdot (\boldsymbol{u}\boldsymbol{u}) - \frac{1}{2}\boldsymbol{u}(\nabla \cdot \boldsymbol{u}) \\ &= \frac{1}{2}[\nabla \cdot (\boldsymbol{u}\boldsymbol{u}) + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u}], \end{aligned} \tag{4}$$

which is skew-symmetric. The advantage of using the skew-symmetric form (4) is that it conserves both the square of velocity as well as the kinetic energy, whereas the divergence form $\nabla \cdot (\boldsymbol{u}\boldsymbol{u})$ conserves only the kinetic energy, and the (original) non-divergence form $(\boldsymbol{u} \cdot \nabla)\boldsymbol{u}$ conserves neither the square of the velocity nor the kinetic energy.

2.2. Initial and boundary conditions

In our investigations, we assume an initial condition

$$\boldsymbol{u}|_{t=0} = \boldsymbol{u}_0(\boldsymbol{x}), \tag{5}$$

that is divergence-free, i.e., $\nabla \cdot \boldsymbol{u}_0 = 0$, and the following boundary conditions

$$\boldsymbol{u}|_{\partial\Omega} = \boldsymbol{u}_b(t),$$

i.e., the velocity is prescribed at the boundary.

Remark. In order to avoid singularities, the initial and boundary conditions are assumed to agree at $t = 0$ and $\boldsymbol{x} \in \partial\Omega$.

The incompressible Navier–Stokes equations can be classified as partial differential–algebraic equations, e.g., [5]. The challenges in their numerical solution are well known; they are connected with the fact that the Navier–Stokes equations are not an evolutionary system of Cauchy–Kovalevskaya type and that the pressure is an implicit function responsible for the satisfaction of the continuity equation. Furthermore, no boundary conditions on the pressure can be imposed on rigid boundaries. This creates formidable obstacles for the construction of fully implicit schemes.