Computers & Fluids 70 (2012) 29–43

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

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

Open boundary conditions for the velocity-correction scheme of the Navier–Stokes equations

A. Poux [a,*], S. Glockner [a], E. Ahusborde [b], M. Azaïez [a]

[a] Université de Bordeaux, IPB-I2M UMR CNRS 5295, 33607 Pessac, France

[b] Laboratoire de Mathématiques et de leurs Applications (U.M.R. 5142 CNRS), Batiment IPRA, Université de Pau et des Pays de l'Adour, France

ARTICLE INFO

Article history:
Received 8 December 2011
Received in revised form 2 July 2012
Accepted 29 August 2012
Available online 21 September 2012

Keywords:
Navier–Stokes equations
Projection methods
Velocity-correction methods
Fractional step methods
Open boundary condition
Pressure boundary condition
Bifurcated tube
Square cylinder
Backward-facing step

ABSTRACT

In this paper we propose to study open boundary conditions for incompressible Navier–Stokes equations, in the framework of velocity-correction methods. The standard way to enforce this type of boundary condition is described, followed by an adaptation of the one we proposed in [36] that provides higher pressure and velocity convergence rates in space and time for pressure-correction schemes. These two methods are illustrated with a numerical test with both finite volume and spectral Legendre methods. We conclude with three physical simulations: first with the flow over a backward-facing step, secondly, we study, in a geometry where a bifurcation takes place, the influence of Reynolds number on the laminar flow structure, and lastly, we verify the solution obtained for the unsteady flow around a square cylinder.

1. Introduction

Efficiently reaching an accurate solution to the unsteady incompressible Navier–Stokes equations is difficult for two main reasons. Firstly, the treatment of non-linearities and secondly, the determination of the pressure field which will ensure a solenoidal velocity field. From all the methods that address this second matter we can sort them in two categories: exact and approximative methods. In the first one, there are all the methods based on the idea proposed by Uzawa et al. [3], like those in [10,16]. In complex geometries or three-dimensional domains, this turns out to be inappropriate since its computational time costs become very high. Augmented Lagrangian is an iterative method described by Fortin and Glowinski in [14]. With this method computing the exact solution is possible but also very costly. Nevertheless an accurate approximation of the solution can be obtained with a small number of iterations. This leads to faster computations but without exactly respecting the incompressibility constraint. The method of interest in this article is one of an another class of non-exact methods which consists in decoupling the pressure from the velocity by means of a time-splitting scheme. This scheme significantly reduces the computational cost of an approximate solution satisfying the incompressibility constraint but with a diminished accuracy.

Since this last class of methods is widely used, a large number of theoretical and numerical works have been published that discuss their accuracy and the stability properties. The state of the art from both theoretical and numerical points of view is described in the review paper of Guermond et al. [20]. The most widespread methods are pressure-correction schemes developed by Chorin, Temam, Goda and later by Timmermans et al. [7,42,17,43]. They require the solution of two sub-steps for each time step. The pressure is treated explicitly in the first step in order to predict a velocity. Then, by projecting the velocity onto an ad hoc space, the solenoidal velocity and the pressure are computed. The governing equation on the pressure or the pressure increment is a Poisson equation derived from the momentum equation by requiring incompressibility. A less studied alternative method known as the velocity correction scheme, developed by Orszag et al. in [33], Karniadakis et al. in [25], Leriche and Labrosse in [26] and more recently by Guermond and Shen in [21], consists in switching the two sub-steps. All these time-splitting schemes have very similar numerical characteristics, but, numerical evidence show that velocity-correction schemes are more stable compared with pressure-correction schemes. This has been reported with high-order time discretization in [25] and with

* Corresponding author. Address: I2M-Trefle, 16 Avenue Pey-Berland, 66307 Pessac, France. Tel.: +33 (0) 540 006 192; fax: +33 (0) 540 006 668.

E-mail addresses: alexandre.poux@enscbp.fr (A. Poux), glockner@enscbp.fr (S. Glockner), etienne.ahusborde@univ-pau.fr (E. Ahusborde), azaiez@enscbp.fr (M. Azaïez).

0045-7930/$ - see front matter
http://dx.doi.org/10.1016/j.compfluid.2012.08.028

Navier–Stokes equations in [11]. In the latter, the authors propose an unconditionally stable scheme with an original implementation of the inertial term.

The majority of the studies based on these methods consider only the Dirichlet boundary condition. However, in many applications such as free surface problems and channel flows, one also has to deal with an outlet boundary condition which should not disturb upstream flow. A large variety of this kind of boundary condition exists [44,39]. Hereafter we will present some results on the open or traction boundary condition which is efficient for low Reynolds number and fluid–structure interactions [27,8,19]. This boundary condition was successfully used to compute various flows such as those around a circular cylinder, over a backward facing step and in a bifurcated tube [27]. Bruneau and Fabrie proposes, in [6], an evolution of the traction boundary condition involving inertial terms.

With open or traction boundary conditions, to our knowledge, several questions remain open specially when a time splitting method is considered. Indeed, while no studies have been reported with a velocity-correction scheme, a few have been done with pressure correction schemes. Guermond et al., have proven in [20] that only convergence rates between one and 3/2 in space and time for velocity and 1/2 in space and time for the pressure are to be expected with the standard incremental scheme. Févrière et al. in [13] combines the penalty and projection methods to offer better error levels. In [36] we presented an almost second-order accurate version of the boundary condition and pressure-correction scheme. We expect to have the same results with velocity-correction schemes as the two are very similar.

The aim of this paper is to study open boundary conditions using the velocity-correction version of the time splitting methods for the incompressible Navier–Stokes equations. In the first part of this article we describe the governing equations, the velocity-correction schemes and the boundary conditions. Since their numerical properties are independent from the treatment of linearities, we only consider in this part Stokes equations. The usual way to enforce this type of boundary condition on the pressure increment is described along with an improvement we proposed in [36] that gives a satisfactory order of convergence for both pressure and velocity. In a second section, we illustrate numerically the behaviour of the standard methods and the proposed method with a manufactured case with both a finite volume and a spectral Legendre method. Finally, in the last section, we study three physical simulations. In the first, we study the flow over a backward-facing step. In the second, we study the influence of the Reynolds number on the laminar flow structure in a geometry where a bifurcation takes place. In the third, we verify the solution obtained for unsteady flow around a square cylinder.

First of all let us specify some notations. Let us consider a Lipschitz domain $\Omega \subset IR^d$, ($d = 2$ or 3), the generic point of Ω is denoted $\boldsymbol{x}$. The classical Lebesgue space of square integrable functions $L^2(\Omega)$ is endowed with the inner product:

$$(\phi, \psi) = \int_\Omega \phi(\boldsymbol{x})\psi(\boldsymbol{x})\, d\boldsymbol{x}$$

and the norm:

$$\|\psi\|_{L^2(\Omega)} = \left(\int_\Omega |\psi(\boldsymbol{x})|^2 d\boldsymbol{x}\right)^{\frac{1}{2}}.$$

We break the time interval $[0, t^*]$ into N subdivisions of length $\Delta t = \frac{t^*}{N}$, called the time step, and define $t^n = n\Delta t$, for any $n, 0 \leqslant n \leqslant N$. Let $\varphi^0, \varphi^1, \ldots, \varphi^N$ be some sequence of functions in $E = L^2$. We denote this sequence by $\varphi^{\Delta t}$, and we define the following discrete norm:

$$||\varphi^{\Delta t}||_{l^2(E)} = \left(\Delta t \sum_{k=0}^{N} ||\varphi^k||_E^2\right)^{\frac{1}{2}} \quad (1.1)$$

In practice the following error estimator can be used:

$$||\varphi||^2_{E(t^*)} = ||\varphi(\cdot, t^*)||_E \quad (1.2)$$

Finally, bold Latin letters like $\boldsymbol{u}, \boldsymbol{w}, \boldsymbol{f}$, etc., indicate vector valued quantities, while capitals (e.g. $\boldsymbol{X}$, etc.) are functional sets involving vector fields.

2. Governing equations

Let Ω be a regular bounded domain in IR^d with $\boldsymbol{n}$ the unit normal to the boundary $\Gamma = \partial\Omega$ oriented outward and $\boldsymbol{\tau}$ the associated unit tangent vector. We assume that Γ is partitioned into two portions Γ_D and Γ_N. Our study consists, for a given finite time interval $[0, t^*]$ in computing velocity $\boldsymbol{u} = \boldsymbol{u}(\boldsymbol{x}, t)$ and pressure $p = p(\boldsymbol{x}, t)$ fields satisfying:

$$\rho\partial_t \boldsymbol{u} - \mu\Delta\boldsymbol{u} + \nabla p = \boldsymbol{f} \quad \text{in } \Omega\times]0, t^*] \quad (2.3)$$
$$\nabla \cdot \boldsymbol{u} = 0 \quad \text{in } \Omega\times]0, t^*] \quad (2.4)$$
$$\boldsymbol{u} = \boldsymbol{g} \quad \text{on } \Gamma_D\times]0, t^*] \quad (2.5)$$
$$(\mu\nabla\boldsymbol{u} - p\boldsymbol{I}) \cdot \boldsymbol{n} = \boldsymbol{t} \quad \text{on } \Gamma_N\times]0, t^*] \quad (2.6)$$

where ρ and μ are respectively the density and the dynamic viscosity of the fluid and $\boldsymbol{I}$ the unit tensor. The body force $\boldsymbol{f} = \boldsymbol{f}(\boldsymbol{x}, t)$, the constraint $\boldsymbol{t} = \boldsymbol{t}(\boldsymbol{x}, t)$ and the boundary condition $\boldsymbol{g} = \boldsymbol{g}(\boldsymbol{x}, t)$ are known. For the sake of simplicity, we chose $\boldsymbol{g} = \boldsymbol{0}$. Finally, the initial state is characterized by a given $\boldsymbol{u}(\cdot, 0)$.

The boundary condition (2.6) is derived from the pseudo-stress tensor $\tilde{\sigma} = \mu\nabla\boldsymbol{u} - p\boldsymbol{I}$. Considering the Cauchy stress tensor $\sigma = \mu(\nabla\boldsymbol{u} + \nabla^T\boldsymbol{u}) - p\boldsymbol{I}$, one can obtain an alternate traction boundary condition containing the non-symmetrical part:

$$\left(\mu\left(\nabla\boldsymbol{u} + \nabla^T\boldsymbol{u}\right) - p\boldsymbol{I}\right) \cdot \boldsymbol{n} = \boldsymbol{t} \text{ on } \Gamma_N\times]0, t^*] \quad (2.7)$$

As we consider the pseudo-stress tensor in (2.3) and later in (2.16), we will only study here the first one for consistency (which is comonly used, see for exemple [27,20]). Nevertheless, a similar study was carried out with the stress tensor and, since the results are very similar, they are not shown here.

2.1. Velocity-correction schemes for open boundary condition

We shall compute two sequences $(\tilde{\boldsymbol{u}}^n)_{0 \leqslant n \leqslant N}$ and $(p^n)_{0 \leqslant n \leqslant N}$ in a recurrent way that approximate in some sense the quantities $(\boldsymbol{u}(\cdot, t^n))_{0 \leqslant n \leqslant N}$ and $(p(\cdot, t^n))_{0 \leqslant n \leqslant N}$, solutions of the unsteady Stokes problem (2.3)–(2.6). The scheme developed by Guermond and Shen (Eqs. (3.6)–(3.8) in [21]) consists of two sub-steps. The first is the prediction problem that computes a pressure increment and a solenoidal velocity: *find* φ^{n+1} and $\boldsymbol{u}^{n+1}$ *such that:*

$$\rho\frac{\alpha\boldsymbol{u}^{n+1} + (\beta - \alpha)\tilde{\boldsymbol{u}}^n + (\gamma - \beta)\tilde{\boldsymbol{u}}^{n-1} - \gamma\tilde{\boldsymbol{u}}^{n-2}}{\Delta t} + \nabla\varphi^{n+1} = \boldsymbol{f}^{n+1} - \boldsymbol{f}^n \quad \text{in } \Omega \quad (2.8)$$
$$\nabla \cdot \boldsymbol{u}^{n+1} = 0 \quad \text{in } \Omega \quad (2.9)$$
$$\boldsymbol{u}^{n+1} \cdot \boldsymbol{n} = 0 \quad \text{on } \Gamma_D \quad (2.10)$$
$$\mu\partial_n(\boldsymbol{u}^{n+1} \cdot \boldsymbol{n}) - p^{n+1} = \boldsymbol{t}^{n+1} \cdot \boldsymbol{n} \quad \text{on } \Gamma_N \quad (2.11)$$

where φ is the pressure increment defined as:

$$\varphi^{n+1} = p^{n+1} - p^n + \chi\mu\nabla \cdot \tilde{u}^n \quad (2.12)$$

The parameter χ is used to switch between the standard incremental scheme ($\chi = 0$) and the rotational one ($\chi = 1$) and parameters α, β, γ depend on the temporal scheme used. Namely:

- $\alpha = 1$, $\beta = -1$, $\gamma = 0$ for the first order Euler time scheme,