Chemical Engineering Science 172 (2017) 585–602

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Numerical solution of the Navier-Stokes equations using the Path Tubes method

Nélio Henderson [a,e,*], Fábio Pacheco [b,e], Mauricio Kischinhevsky [c,d]

[a] Instituto Politécnico, Universidade do Estado do Rio de Janeiro, 28601-970 Nova Friburgo, Rio de Janeiro, Brazil
[b] Instituto do Norte Fluminense de Educação Superior, Universidade Federal Fluminense, 28470-000 Santo Antônio de Pádua, Rio de Janeiro, Brazil
[c] Instituto de Computação, Universidade Federal Fluminense, 24210-346 Niterói, Rio de Janeiro, Brazil
[d] Dept. of Computer Science, University of British Columbia, V6T 1Z4 Vancouver, B.C., Canada
[e] Thermodynamics and Optimization Group (TOG), Brazil

HIGHLIGHTS

- We extend the Path Tubes method to the Navier-Stokes equations.
- The method use an approach based on the mechanics of continuous media.
- Were used values of Reynolds numbers typical of advection dominated flows.
- The method proved to be accurate for the calculation of velocity fields.
- The proposed methodology is able to work with coarse grids.

ARTICLE INFO

Article history:
Received 12 January 2017
Received in revised form 8 June 2017
Accepted 7 July 2017
Available online 10 July 2017

Keywords:
Path Tubes method
Semi-Lagrangian algorithm
Navier-Stokes equation
Implicit scheme
Interpolation formula
Advection-dominated flow

ABSTRACT

The present work addresses an extension of the Path Tubes method for solving the time-dependent Navier-Stokes equations for an incompressible Newtonian fluid. The approach used is a physically intuitive methodology whose formulation is based on the theoretical foundations of the mechanics of continuous media. This version of the Path Tubes method draws on a semi-Lagrangian time discretization that employs the Reynolds's transport theorem and a localization strategy. This time discretization can be seen as a transformation that acts on the Navier-Stokes equations, transforming this classical nonlinear model into linear partial differential equations of the (essentially) parabolic type.

The result is an implicit semi-Lagrangian algorithm that allows the use of classical schemes for spatial discretization such as central-difference formulas, without the need to use upwind techniques or high-order corrections for time derivatives. The Path Tubes method was implemented through parallel computing. For this, we use a computer equipped with shared-memory multiprocessors and the OpenMP software. After intensive numerical tests and using different values of Reynolds numbers typical of advection-dominated flows, the proposed method proved to be accurate and able to work with coarse grids.

1. Introduction

The displacement of an incompressible Newtonian fluid is governed by the Navier-Stokes equations, which are second-order nonlinear partial differential equations. Due to the relevance of these equations in the simulation of different phenomena and processes of importance to the applied sciences and engineering, the development of numerical techniques for solving the Navier-Stokes equations has attracted the interest of many researchers, still being one of the central aspects of the research area commonly called computational fluid dynamics (CFD) (Gresho and Sani, 2000; Ferziger and Perić, 2002).

A numerical approach for the time-dependent Navier-Stokes equations must take into account their description in time and space, in addition to an appropriate treatment of the nonlinearities.

The unsteady and advective terms of the momentum equations bring to the Navier-Stokes equations a non-linear hyperbolic character, while the unsteady and diffusive (viscous) terms of these equations are similar to their counterparts in a typical linear parabolic partial differential equation, such as the heat conduction

* Corresponding author at: Instituto Politécnico, Universidade do Estado do Rio de Janeiro, 28601-970 Nova Friburgo, Rio de Janeiro, Brazil.
E-mail address: nelio@iprj.uerj.br (N. Henderson).

http://dx.doi.org/10.1016/j.ces.2017.07.016
0009-2509/

equation. Thus, depending on the value of the so-called Reynolds number, one of these characteristics may prevail over the other, making the numerical treatment of the Navier-Stokes equations likely to be a complex problem.

No greater difficulties are found in solving the Navier-Stokes equations at low Reynolds number, employing different numerical approaches typically used in the discretization of parabolic equations, such as central-difference approximations. However, at high Reynolds number, where the advective phenomenon is dominant, these equations are difficult to solve (Ghia et al., 1982). Particularly in a semi-implicit Eulerian method, where advection is treated explicitly, time step sizes are subject to a stability condition (Bulgarelli et al., 1984). Generally, such a stability condition imposes severe restrictions on the time step size. Thus, this constraint may make a semi-implicit Eulerian scheme prohibitive at high Reynolds numbers. This is exactly what happens when centered finite differences are used to discretize the convective terms (Casulli, 1988). Usually, fully implicit Eulerian methods for Navier-Stokes equations are unconditionally stable (Ferziger and Perić, 2002). Many of these fully implicit methods require a very thin discretization grid, which is computationally expensive.

As initially verified by Pironneau (1982), a semi-Lagrangian time discretization can be a good alternative for the numerical treatment of the Navier-Stokes equations.

Actually, semi-Lagrangian methods are part of the historical legacy of computational fluid dynamics. In fact, the standard approach to numerical modeling of advection phenomena in the 1950s and 1960s was the Lagrangian treatment (Wiin-Nielsen, 1959; Trulio, 1965; Hirt et al., 1970).

Because an advantage of these semi-Lagrangian methods lies in the fact that they are usually not subject to restrictions imposed by stability conditions, historically such methodologies have long been used for numerical computation of advection equations when large time steps become necessary, as in meteorology for numerical weather prediction (Robert, 1981; Staniforth and Côté, 1991; Smolarkiewicz and Pudykiewicz, 1992).

The main drawback of most semi-Lagrangian schemes is that they formally do not conserve mass (or other conservative properties), which can compromise the accuracy of numerical solutions generated by these methods. Recent works have led to improvements in the conservative property of semi-Lagrangian algorithms for advection equations (Rančić, 1992; Gravel and Staniforth, 1994; Laprise and Plante, 1995; Nakamura et al., 2001; Yabe et al., 2001, 2002; Bermejo and Conde, 2002; Nair et al., 2003; Sun and Sun, 2004; Henderson et al., 2009, 2011).

Since the pioneering work of Pironneau (1982), different authors have applied semi-Lagrangian approaches to the Navier-Stokes equations (Huffenus and Khaletzky, 1984; Süli, 1988; Casulli, 1988; Buscaglia and Dari, 1992; Boukir et al., 1994, 1997; Allievi and Bermejo, 2000; Xiu and Karniadakis, 2001; Benitez and Bermudez, 2015; Celledoni et al., 2015; Piao et al., 2015).

The Path Tubes method is a semi-Lagrangian algorithm introduced by Henderson et al. (2009, 2011) for advection equations whose formulation is based on the foundations of the mechanics of continuous media. This semi-Lagrangian algorithm is physically intuitive and uses the so-called Reynolds's transport theorem to judiciously establish its main property on the basis of conservative integral equations. In the present work, we extend the application of the Path Tubes method to the Navier-Stokes equations.

The proposed methodology has the property of integrating the Navier-Stokes equations along space-time so that (after the use of the Reynolds theorem and a localization procedure) the advective terms are no longer part of the resulting partial differential equations. In fact, the resulting equations have an essentially parabolic form equipped with a semi-Lagrangian time discretization for the material derivatives. This approach allows the use of classical schemes for spatial discretization, such as the central-difference formulas employed here, without the need to use upwind techniques or high-order corrections for time derivatives.

To determine values at upstream points, many semi-Lagrangian methods use backtracking processes in time. Because the calculated downstream points may not coincide with computational grid points, semi-Lagrangian methods generally depend on interpolations of initial profiles (Xiao, 2000). The choice of the interpolation formula can have an impact on the accuracy of the numerical solution.

In this study, we use a simplified version of the inverse distance-weighted interpolation formula. This numerical technique is a well-established interpolation formula that was originally developed by Shepard (1968) for use in the development of geographic information systems.

As shown in previous tests conducted by Henderson et al. (2009, 2017), this simplified version of Shepard's interpolation formula is appropriate to equip the Path Tubes method.

In the form considered here, the Path Tubes method uses the primitive variables (velocity components and pressure), to calculate the velocity field, solving a linear system with infinitely many solutions for the pressure.

To solve this linear system, we used an iterative algorithm that does not require the computation and/or storage of matrices, which was implemented using parallel computing via the OpenMP software (Chandra et al., 2001).

To test the numerical performance of the proposed methodology for the Navier-Stokes equations, we employ two-dimensional benchmark problems available in the literature.

Our results show that the Path Tubes method is competitive with the state-of-the-art numerical techniques for the solution of the Navier-stokes equations, such as finite difference, finite volume and finite element methods.

The rest of the paper is organized as follows. In Section 2, we describe the Path Tubes method for Navier-Stokes equations. Section 3 is devoted to the interpolation formula. The iterative method used to solve the resulting system of the space-time discretization is described in Section 4. In Section 5, we report the numerical results. The conclusions are given in Section 6.

2. The path tubes method for the Navier-Stokes equations

2.1. Basic aspects

The Path Tubes method is based on basic aspects of continuous mechanics, in light of a semi-Lagrangian methodology (Henderson et al., 2009, 2011). In view of this, we initially consider some physical concepts of interest related to the basic kinematics of continuous media and some notations used in this work.

The scalar product of vectors v_1 and v_2 in $\mathbb{R}^n$ is denoted by $v_1.v_2$. We use $\|v\|$ to denote the Euclidean norm of $v \in \mathbb{R}^n$, i.e., $\|v\| = \sqrt{v.v}$.

$\mathfrak{B}$ denotes a material body, and $\mathbb{E} = \mathbb{R}^n$ is an n-dimensional Euclidean space, where $n \in \{1, 2, 3\}$. Given a time $t \in \mathbb{R}$, the position occupied by $\mathfrak{B}$ in $\mathbb{E}$ (at time t) will be denoted by $\Omega_t \subset \mathbb{E}$, where Ω_t is a regular region of $\mathbb{E}$, which is called the configuration of the material body $\mathfrak{B}$ at time t. A material body is characterized by the possibility of occupying different regions $\Omega_t \subset \mathbb{E}$ at different times. However, as emphasized by Gurtin (1981), none of these regions can be intrinsically associated with the body. To address this issue, we select Ω_{t_R}, the configuration of $\mathfrak{B}$ at a particular instant t_R. Thus, a given material particle of $\mathfrak{B}$ will be associated with a single material point $\xi \in \Omega_{t_R}$.Continuum mechanics is, essentially, a study of deforming bodies. Mathematically, at a given time t, the body $\mathfrak{B}$ is deformed via a one-to-one mapping