Available online at www.sciencedirect.com
SCIENCE DIRECT®
Journal of Computational Physics 203 (2005) 49–71
JOURNAL OF COMPUTATIONAL PHYSICS
www.elsevier.com/locate/jcp

High order accurate solution of the incompressible Navier–Stokes equations

Arnim Brüger [a], Bertil Gustafsson [b], Per Lötstedt [b,*], Jonas Nilsson [b]

[a] *Department of Mechanics, Royal Institute of Technology, SE-10044 Stockholm, Sweden*
[b] *Department of Information Technology, Scientific Computing, Uppsala University, P.O. Box 337, SE-75105 Uppsala, Sweden*

Received 19 December 2003; received in revised form 17 June 2004; accepted 3 August 2004
Available online 13 October 2004

Abstract

High order methods are of great interest in the study of turbulent flows in complex geometries by means of direct simulation. With this goal in mind, the incompressible Navier–Stokes equations are discretized in space by a compact fourth order finite difference method on a staggered grid. The equations are integrated in time by a second order semi-implicit method. Stable boundary conditions are implemented and the grid is allowed to be curvilinear in two space dimensions. The method is extended to three dimensions by a Fourier expansion. In every time step, a system of linear equations is solved for the velocity and the pressure by an outer and an inner iteration with preconditioning. The convergence properties of the iterative method are analyzed. The order of accuracy of the method is demonstrated in numerical experiments. The method is used to compute the flow in a channel, the driven cavity and a constricted channel.

MSC: 65N50; 76M12
Keywords: Finite difference method; High order; Incompressible flow; Iterative solution; Curvilinear coordinates

1. Introduction

Spectral and pseudospectral methods are accurate methods for direct numerical simulation of turbulent flow governed by the incompressible Navier–Stokes equations. The disadvantage with these methods is that they are restricted to simple geometries such as channels. Finite difference methods of high order do not have this restriction and are almost as accurate as a spectral method. In this paper we develop such

* Corresponding author. Tel.: +46 18 471 2972; fax: +46 18 523049.
E-mail address: perl@it.uu.se (P. Lötstedt).

0021-9991/$ - see front matter © 2004 Elsevier Inc. All rights reserved.
doi:10.1016/j.jcp.2004.08.019

a method of fourth order accuracy in space and second order in time for two-dimensional problems (2D) and indicate how the method is extended to three dimensions (3D). The level of the discretization error is the important issue, not so much the order of accuracy, but a certain error level is more easily obtained with a high order method. High order is the means to reach the goal: a small discretization error in the solution. The same accuracy is achieved with fewer grid points or the solution has better accuracy on the same grid compared to a second order method. The solution in curvilinear body-fitted coordinates is obtained by a mapping of the equations to a rectangular grid where the derivatives are approximated. The solution is computed in the primitive variables defined on a staggered grid with local velocity components to avoid spurious oscillations. High order accuracy in the spatial approximations is also necessary for large eddy simulation (LES) in order not to interfere with the subgrid model with terms proportional to the square of the grid size [17]. It is not necessary for LES to have high order also in the time discretization. The error caused by the time stepping can be controlled independently by changing the length of the time step.

Let u and v be the velocity components in the x- and y-direction, respectively, p the pressure, and ν the kinematic viscosity. The Reynolds number is defined by $Re = u_b\ell/\nu$ for some characteristic velocity u_b and length scale ℓ. By defining $\boldsymbol{w} = (u\,v)^T$, differentiation with respect to an independent variable, such as time t, by a subscript, and the nonlinear and linear terms

$$\mathscr{N}(\mathbf{w}) = (\mathbf{w}\cdot\nabla)\mathbf{w}, \quad \mathscr{L}(\mathbf{w},p) = \nabla p - Re^{-1}\Delta\mathbf{w},$$

the Navier–Stokes equations in two dimensions are

$$\mathbf{w}_t + \mathscr{N}(\mathbf{w}) + \mathscr{L}(\mathbf{w},p) = 0, \tag{1}$$

$$\nabla\cdot\mathbf{w} = 0. \tag{2}$$

The space discretization of $\mathscr{N}$ and $\mathscr{L}$ in (1) is in compact form [27] and the equations are integrated in time by a semi-implicit method. The approximations of the first and second derivatives are defined implicitly and they satisfy systems of linear equations with a tridiagonal system matrix. The number of terms in the computational domain is reduced by orthogonal grids in the physical domain. The nonlinear convection term $\mathscr{N}$ is extrapolated from the previous time steps. The linear part of the discretized momentum equations $\mathscr{L}$ and the discretized divergence equation (2) are solved simultaneously at the new time level. A system of linear equations has to be solved for $\boldsymbol{w}$ and p in every time step. The solution is computed by an iterative method until convergence is reached. The method is shown to be of fourth order in space and of second order in time in $\boldsymbol{w}$ and p in numerical experiments. The method is analyzed with respect to boundary conditions and stability in [20,21,26,35]. Other examples of the accuracy and performance of the method are found in [4,6].

The difference scheme approximates the Navier–Stokes equations in its original form including the continuity equation and well-posed boundary conditions. In this way the usual difficulty of forming boundary conditions for the pressure equation is avoided [3,42]. The solution procedure for the complete algebraic system of equations uses a factorization that requires the solution of a subsystem that corresponds to a discrete elliptic equation of Poisson type. However, the boundary conditions for this equation are automatically obtained as part of the factorization.

There are stability constraints on the time step Δt due to the semi-implicit time integration. Other constraints on Δt are introduced by accuracy requirements and the convergence of the iterative solvers. It is shown in [9] that an implicit method is faster than a semi-implicit method for fully developed turbulent flow in a boundary layer but the CFL-number for good accuracy is rather low, 0.5–1. In other flow regimes, such as transitional flow, a much smaller time step is necessary for accuracy [15], thus reducing the time savings with an implicit method.

The method is extended to three space dimensions aiming at direct numerical simulation of turbulent flow. The solution is assumed to be periodic in the spanwise direction and the variables are expanded in