Journal of Computational Mathematics, Vol.18, No.5, 2000, 521–540.

NUMERICAL SOLUTION OF THE UNSTEADY INCOMPRESSIBLE NAVIER–STOKES EQUATIONS ON THE CURVILINEAR HALF–STAGGERED MESH*1)

Lan-chieh Huang
(*Institute of Computational Mathematics and Scientific/Engineering Computing, Chinese Academy of Sciences, Beijing 100080, China*)

Abstract

In this paper, the Crank–Nicholson + component–consistent pressure correction method for the numerical solution of the unsteady incompressible Navier–Stokes equation of [1] on the rectangular half–staggered mesh has been extended to the curvilinear half–staggered mesh. The discrete projection, both for the projection step in the solution procedure and for the related differential–algebraic equations, has been carefully studied and verified. It is proved that the proposed method is also unconditionally (in Δt) nonlinearly stable on the curvilinear mesh, provided the mesh is not too skewed. It is seen that for problems with an outflow boundary, the half–staggered mesh is especially advantageous. Results of preliminary numerical experiments support these claims.

Key words: Unsteady incompressible Navier-Stokes equations, Curvilinear half–staggered mesh, Discrete projection.

1. Introduction

Let us consider the unsteady incompressible Navier–Stokes equations (INSE)

$$\frac{\partial \mathbf{w}}{\partial t} + (\mathbf{w} \cdot grad)\mathbf{w} + grad\ p = \frac{1}{Re} div\ grad\ \mathbf{w} \tag{1.1}$$

$$div\ \mathbf{w} = 0 \tag{1.2}$$

on a two–dimensional region Ω with boundary $\partial\Omega$. Here $\mathbf{w}$ is the velocity vector; in terms of its Cartesian components $\mathbf{w} = (u, v)^T$; p is the pressure. The initial condition is given as

$$\mathbf{w}|_{t=0} = \mathbf{w}^0 \quad \text{on } \Omega \tag{1.3}$$

satisfying (1.2). We are concerned mainly with the solid wall boundary condition

$$\mathbf{w} = \mathbf{w}_B \quad \text{on } \partial\Omega \quad \text{satisfying } \oint_{\partial\Omega} w_n ds = 0 \tag{1.4}$$

* Received November 19, 1997.
1) Supported by Projects 19472068 and 19772056 of the National Natural Science Foundation of China and the Laboratory of Scientific and Engineering Computing of the Institute of Computational Mathematics, the Chinese Academy of Sciences.

but we will also briefly discuss the outflow boundary condition. Note the convection term can also be written in conservative form $(\mathbf{w} \cdot grad)\mathbf{w} = div(\mathbf{ww})$, using (1.2).

The difficulty in the numerical solution of the above problem lies in that (1.1) and (1.2) are partial differential equations with constraint; i.e. the system of equations is not entirely evolutionary. The projection methods of [2], [3], and [4] have been widely used and have proven to be most efficient for this type of problems. However, it is not always very well understood and has caused a great deal of confusion, e.g. see [5], [6], and [7] for discussions of the numerical boundary layers in pressure. For spatial discretization, there are many finite element methods – the so–called mixed methods. But for high *Re* unsteady complex flow simulation, the finite difference methods are usually used. It may be surprising to those not in the immediate field that the finite difference discretization is almost exclusively done on the staggered mesh of [8] for practical computation. The reason is that on the half–staggered mesh or the general mesh, the centered difference scheme (for $grad\ p$ and $div\ \mathbf{w}$) is not "regular" and the numerical solution is not "smooth", see [9] and [10], mostly the pressure solution can be intractable.

Because many of the advantages of the staggered mesh are lost on its curvilinear counterpart, see [11], we have directed our efforts toward the half–staggered mesh of [12], see Fig. 1. This mesh retains some of the advantages of the staggered mesh and does not need half–interval differencing on points adjacent to the boundary. Its main advantage lies in that with both components of the velocity at the same point, their coordinate transformation, the discretization of (1.2), and the formation of boundary conditions become more intelligible. But the solution of the discrete Poisson equation for pressure (or pressure correction) in the projection step becomes troublesome, in that there is an added constraint for solution and in that the solution can have oscillations, see [13] and [14]. We have shown in [15] that the added constraint is of no serious consequences for many problems, and that the oscillations do not affect the discrete gradient of p, which is of our only concern. Furthermore, for simulation of high *Re* unsteady complex flow in rectangular regions, a fast solver for the discrete Poisson equation with the most straightforward finite difference approximation of *grad* and *div* on the half–staggered mesh has been developed in [16]. It has proven to be very efficient in the numerical tests of [17] and [1]. We point out here that the half–staggered mesh has been used quite successfully in [18], but with a different procedure for projection, in which the divergence–free velocity is computed directly with a Galerkin approach.

We have chosen the pressure correction (PC) projection method of [4] because its equation for the auxiliary velocity is consisitent with (1.1), and hence the boundary condition (1.4) can be used. Also it retains the second order time accuracy of the underlying difference scheme, say the Crank–Nicolson (CN) scheme. However with the regular PC projection method, the "deviation" problem is sometimes encountered in practical computation. In [1], it is explained that with spatial discretization on a *fixed* mesh, the INSE becomes a system of differential–algebraic equations (DAE). For its solution to evolve along the correct branch, a consistency condition between the components (here $\mathbf{w}$ and p) of the solution must be satisfied. The PC projection method does not preserve this consisitency condition, and hence may lead to "deviation"