CHAPTER

TWELVE

THE NAVIER-STOKES EQUATION

Numerical methods have now advanced to the stage whereby a wide variety of phenomona can be simulated that in the past could only be studied experimentally. Fluid mechanics has traditionally been divided into groups such as laminar or turbulent flow, while laminar flow has been subdivided into slow flow situations (Stokes flow) and high speed situations (leading to boundary layers, which require special boundary layer methods). Stokes flow applies for zero Reynolds numbers; in this situation, analytic solutions can occasionally be obtained. As the Reynolds number increases, however, numerical solutions are required. Boundary layer methods are useful at high speeds (large Reynolds numbers) when a thin boundary layer develops; in a few situations analytic solutions are possible. Numerical methods can be used to bridge the gap between a Reynolds number of zero and a large Reynolds number. In addition, numerical methods allow the solution of cases with more complicated boundary conditions than can usually be handled analytically, even for a zero Reynolds number. As the Reynolds number increases to the point that boundary layer methods are applicable, the numerical method must use appropriate mesh refinement in the boundary layer or change to a boundary layer method. We can regard the tools available to solve the Navier-Stokes equation for laminar flow as essentially well developed. Turbulent flow is modeled using equations that have their parameters determined by experiment. Usually these equations are not fundamental ones, in that they are not derivable from first principles, but they are useful nonetheless. It is possible to directly simulate turbulence, but the mesh must be very small and the calculations are time-consuming and thus very expensive. In this book only laminar flows are discussed. In this chapter, we will study both finite element methods and finite difference methods for solving the Navier-Stokes equation.

12.1. Equations

The Navier-Stokes equation and the energy equation are listed in Table 12.1. A few assumptions have been made:

Table 12.1. Equations

Navier-Stokes Equations:

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho\, \mathbf{u} \cdot \nabla\ \mathbf{u} = \rho\, \mathbf{f} - \nabla p + \nabla \cdot [\mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)]$$

Energy Equation:

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) = \nabla \cdot (k \nabla T) + \Phi_v$$

Navier-Stokes Equations in Cartesian Coordinates (2D):

$$\rho\left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = \rho f_x - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(2\mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right]$$

$$\rho\left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = \rho f_y - \frac{\partial p}{\partial x} + \frac{\partial}{\partial y}\left(2\mu \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right]$$

Energy Equation in Cartesian Coordinates (2D):

$$\rho C_p \left(\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} \right) = \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y}\left(k \frac{\partial T}{\partial y} \right) + \Phi_v$$

1. Newton's law of motion
2. Law of conservation of energy
3. Incompressible fluid
4. Continuum
5. Newtonian fluid

No two-dimensional or three-dimensional experiment should be used to test these assumptions; these assumptions are either axioms or can be tested with simple experiments. Newton's law and the law of conservation of energy are axioms, which we accept unless relativistic effects are important. The incompressibility of a fluid can be tested by measurement or by looking up a compressibility values in a table. Indeed, if desired, simulations can be done for compressible fluids. Whether or not a fluid is a continuum can be decided based on kinetic theory and the perfect gas law. The fifth assumption, a Newtonian fluid, is an important one. We can measure a fluid's viscosity in a viscometer and determine if the shear stress is directly proportional to the shear rate and if elastic effects are absent. If so, the fluid is Newtonian and it will be discussed in this chapter; if not, it will be discussed in Chapter 13.

Let us further simplify the equations by also assuming the following:

6. Two-dimensional flow
7. Laminar flow

The Navier-Stokes equation is written in Cartesian coordinates in Table 12.1. It is important to do an experiment (or to read literature where experimental information is reported) to determine whether or not the flow is two-dimensional and laminar. If the flow is two-dimensional and laminar, then it is easier, faster, and cheaper to solve the Navier-Stokes equation numerically than it is to measure some aspect of the flow. This statement could not have been made twenty years ago, but it proves true today. The most important part of any model (or experiment) is how the system interacts with its surroundings, i.e., the boundary conditions. Thus an experiment may be necessary to show that the boundary conditions of the theory are appropriate or that an important phenomenon is not left out. On the other hand, calculations can be used to assess the validity of assumptions made in interpreting the experimental results.

12.2. Finite Element Methods for Steady-State Problems

Here we concentrate on two-dimensional, laminar flows. A finite element method can be applied to the Navier-Stokes equation in a variety of ways:

1. Primitive variables (velocity and pressure)
2. Primitive variables (velocity) with pressure from a penalty method
3. Stream function-vorticity

In the first and second cases, the velocity variables are expanded in a series and the decision between the two cases is essentially an economic one (for Newtonian fluids). In the third case, the primary variables are derivatives or integrals of velocity; sometimes the boundary conditions are difficult to specify because the variable of interest is not a physical quantity. Finite element methods applied to the Navier-Stokes equation are described in detail in a book by Cuvelier, *et al.* [1986].

Let us first consider the case when both velocity and pressure are expanded in a series:

$$u = \sum_{i=1}^{NU} u_i N_i(x,y), \quad v = \sum_{i=1}^{NU} v_i N_i(x,y), \quad p = \sum_{i=1}^{NP} p_i N_i'(x,y). \tag{12.1}$$

Typically the velocity is expanded using an order that is one higher than the pressure. Thus the velocity may be quadratic and the pressure linear or the velocity may be linear and the pressure constant. The reason different degree polynomials are used for pressure and velocity is so that the discrete equations will have a solution for pressure when the velocity is completely specified (Jackson and Cliffe [1981], Sani, *et al.* [1981]). The trial function may be defined on triangles or quadrilaterals with constant, linear, or quadratic interpolation.

Next we put the trial function into the Navier-Stokes and continuity equations to define the residuals. To illustrate the procedure, let us use the equations in non-dimensional form:

$$\mathrm{Re}\,\mathbf{u}\cdot\nabla\mathbf{u} = -\nabla p + \nabla\cdot[\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)], \tag{12.2}$$

$$\nabla\cdot\mathbf{u} = 0 \tag{12.3}$$

with the boundary conditions

$$\begin{aligned} &\mathbf{n}\cdot\mathbf{u} = u_n \text{ or } \mathbf{n}\cdot\boldsymbol{\tau}\cdot\mathbf{n} = -p + \mu\,\mathbf{n}\cdot[\nabla\mathbf{u} + \nabla\mathbf{u}^T]\cdot\mathbf{n} = f_n, \\ &\mathbf{t}\cdot\mathbf{u} = u_t \text{ or } \mathbf{n}\cdot\boldsymbol{\tau}\cdot\mathbf{t} = \mu\,\mathbf{n}\cdot[\nabla\mathbf{u} + \nabla\mathbf{u}^T]\cdot\mathbf{t} = f_t, \end{aligned} \tag{12.4}$$

where the Reynolds number is $\rho u_s x_s/\mu_s$, with u_s, x_s, and μ_s as reference quantities representing an average velocity, a certain characteristic length, and a viscosity. Here the symbol μ refers to the actual viscosity (which may depend on temperature) divided by the reference viscosity (a constant). The momentum residuals are multipled by $\delta\mathbf{u}$ (which is $N_j\mathbf{e}_x$ or $N_j\mathbf{e}_y$) and integrated over the domain, giving

$$\mathrm{Re}\int_V \delta\mathbf{u}\cdot(\mathbf{u}\cdot\nabla\mathbf{u})\,dV = -\int_V \delta\mathbf{u}\cdot\nabla p\,dV + \int_V \delta\mathbf{u}\cdot\{\nabla\cdot[\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)]\}dV. \tag{12.5}$$

The viscous terms are integrated by parts and the divergence theorem is applied, giving

$$\begin{aligned} \int_V \delta\mathbf{u}\cdot\{\nabla\cdot[\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)]\}dV &= \int_V \nabla\cdot\{\delta\mathbf{u}\cdot[\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)]\}dV - \\ &\int_V \nabla\delta\mathbf{u} : [\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)]\}dV \\ = \int_S \{\delta\mathbf{u}\cdot[\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)]\}\cdot\mathbf{n}\,dS &- \int_V \nabla\,\delta\mathbf{u} : [\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)]\}dV \end{aligned} \tag{12.6}$$

The pressure term is also integrated by parts and the divergence theorem is used:

$$\begin{aligned} \int_V \delta\mathbf{u}\cdot\nabla p\,dV &= \int_V \nabla\cdot(\delta\mathbf{u}\,p)dV - \int_V p\,\nabla\cdot\delta\mathbf{u}\,dV \\ &= \int_S \mathbf{n}\cdot\delta\mathbf{u}\,p\,dS - \int_V p\,\nabla\cdot\delta\mathbf{u}\,dV \end{aligned} \tag{12.7}$$

The complete result is then

$$\begin{aligned} \mathrm{Re}\int_V \delta\mathbf{u}\cdot(\mathbf{u}\cdot\nabla\mathbf{u})\,dV &= \int_V p\,\nabla\cdot\delta\mathbf{u}\,dV - \int_V \nabla\delta\mathbf{u} : [\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)]\}dV - \\ -\int_S \mathbf{n}\cdot\delta\mathbf{u}\;p\,dS &+ \int_S \{\delta\mathbf{u}\cdot[\mu(\nabla\mathbf{u} + \nabla\mathbf{u}^T)]\}\cdot\mathbf{n}\,dS \end{aligned} \tag{12.8}$$

We would not use this equation for a node on the boundary for which the velocity must take a specified value. In the last two terms we separate the velocity into its normal and tangential components.

$$\delta \mathbf{u} = \mathbf{n} \cdot \delta \mathbf{u} + \mathbf{t} \cdot \delta \mathbf{u}\, \mathbf{t} \tag{12.9}$$

Then we substitute the following into the last two terms of Eq. (12.8):

$$-\int_S \mathbf{n} \cdot \delta \mathbf{u}\; p\, dS + \int_S \{ \delta \mathbf{u} \cdot [\mu (\nabla \mathbf{u} + \nabla \mathbf{u}^T)]\} \cdot \mathbf{n}\, dS =$$

$$\int_S \mathbf{n} \cdot \delta \mathbf{u}\; [f_n - \mathbf{n} \cdot \mu (\nabla \mathbf{u} + \nabla \mathbf{u}^T) \cdot \mathbf{n}]\, dS +$$

$$+ \int_S \mathbf{n} \cdot \delta \mathbf{u}\; \mathbf{n} \cdot \mu (\nabla \mathbf{u} + \nabla \mathbf{u}^T) \cdot \mathbf{n}\; dS + \int_S \mathbf{t} \cdot \delta \mathbf{u}\, f_t\, dS \tag{12.10}$$

to give

$$\mathrm{Re} \int_V \delta \mathbf{u} \cdot (\mathbf{u} \cdot \nabla \mathbf{u})\, dV = \int_V p \nabla \cdot \delta \mathbf{u}\, dV - \int_V \nabla \delta \mathbf{u} : [\mu (\nabla \mathbf{u} + \nabla \mathbf{u}^T)]\} dV +$$
$$+ \int_{S_1} \mathbf{n} \cdot \delta \mathbf{u}\, f_n\, dS + \int_{S_2} \mathbf{t} \cdot \delta \mathbf{u}\, f_t\, dS \tag{12.11}$$

The essential boundary conditions are velocities specified on the boundary. If the normal velocity is specified, then we do not use Eq. (12.11) for the normal component of velocity or use the integral over S_1. Similar considerations apply to the tangential component of velocity and the integral over S_2. Since the essential boundary conditions must be applied to the normal and tangential components of velocity, it is clearly useful to have the flow boundaries parallel to the coordinate axes if an essential boundary condition is used for the normal direction and a natural boundary condition is used in the tangential direction (and vice versa). Finally, the continuity equation is multiplied by N_j', the trial function for pressure. The continuity equation is essentially an equation for pressure; furthermore, this treatment gives the same result as is given by the variational principle for slow flow.

$$\int_V \delta p \nabla \cdot \mathbf{u}\, dV = 0 \tag{12.12}$$

We then solve Eq. (12.11) and Eq. (12.12). These represent a set of nonlinear algebraic equations that must be solved iteratively. The most common way that these equations are solved is by using the Newton-Raphson method: the equations are linearized about the current iterate and only the first-order terms are kept. Denoting the iteration number with a superscript then gives

$$\mathrm{Re} \int_V \delta \mathbf{u} \cdot (\mathbf{u}^{n+1} \cdot \nabla \mathbf{u}^n)\, dV + \mathrm{Re} \int_V \delta \mathbf{u} \cdot (\mathbf{u}^n \cdot \nabla \mathbf{u}^{n+1})\, dV - \int_V p^{n+1} \nabla \cdot \delta \mathbf{u}\, dV +$$

$$+ \int_V \nabla \delta \mathbf{u} : [\mu (\nabla \mathbf{u}^{n+1} + \nabla \mathbf{u}^{n+1,T})]\} dV +$$

$$= -\mathrm{Re} \int_V \delta \mathbf{u} \cdot (\mathbf{u}^n \cdot \nabla \mathbf{u}^n)\, dV + \int_{S_1} \mathbf{n} \cdot \delta \mathbf{u}\, f_n\, dS + \int_{S_2} \mathbf{t} \cdot \delta \mathbf{u}\, f_t\, dS \tag{12.13}$$

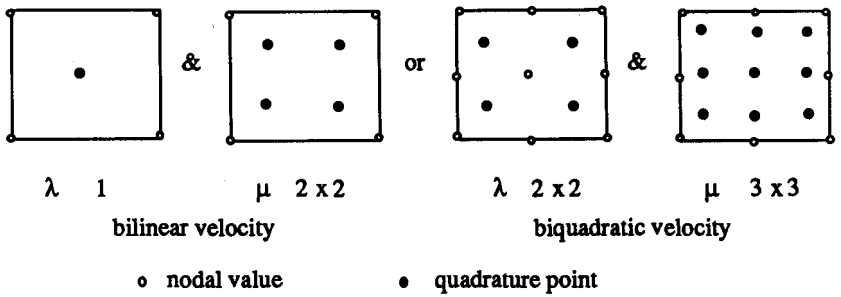

Figure 12.1. Quadrature Points to Use with the Penalty Method

An initial guess is supplied for u^0, v^0, and p^0. Then Eq. (12.13) is applied once to find u^1, v^1, and p^1; this process is repeated until the successive iterates are within some error tolerance (such as 10^{-6} when the calculations are done in double precision). If the Reynolds number is small (e.g., 0.00001), only one iteration is necessary. If the Reynolds number is 100, usually only four or five iterations are necessary if a reasonable initial guess is provided. For this reason, a good strategy is to always solve the case for Re = 0 first and use that as the initial guess for larger Reynolds numbers. If the Reynolds number is 1000 or more, then the method may not converge at all, depending on the initial guess and the mesh. One important point is that the pressure must be specified for at least one node, since the differential (and algebraic) equations are indeterminant to within a constant pressure.

In the penalty method, the incompressibility condition is handled as a Lagrange multiplier. We write the pressure equation as

$$p = -\lambda \nabla \cdot \mathbf{u}, \tag{12.14}$$

where λ is a large constant, chosen by the analyst. Then Eq. (12.13) becomes

$$\begin{aligned} &\mathrm{Re}\int_V \delta\mathbf{u}\cdot(\mathbf{u}^{n+1}\cdot\nabla\,\mathbf{u}^n)\,dV + \mathrm{Re}\int_V \delta\mathbf{u}\cdot(\mathbf{u}^n\cdot\nabla\,\mathbf{u}^{n+1})\,dV + \lambda\int_V(\nabla\cdot\mathbf{u}^{n+1})\,\nabla\cdot\delta\mathbf{u}\,dV + \\ &\quad + \int_V \nabla\,\delta\mathbf{u} : [\,\mu\,(\nabla\,\mathbf{u}^{n+1} + \nabla\,\mathbf{u}^{n+1,T})]\}dV + \\ &= -\mathrm{Re}\int_V \delta\mathbf{u}\cdot(\mathbf{u}^n\cdot\nabla\,\mathbf{u}^n)\,dV \int_{S_1}\mathbf{n}\cdot\delta\mathbf{u}\,f_n\,dS + \int_{S_2}\mathbf{t}\cdot\delta\mathbf{u}\,f_t\,dS \end{aligned} \tag{12.15}$$

We thus have to solve for fewer unknowns since the only unknown values are the velocities u_i and v_i. If we want the pressure, we can calculate it using Eq. (12.14), although smoothing is necessary. However, there is a problem. For a Reynolds number of zero if we solve the problem as stated, the solution converges to the zero solution (i.e., zero velocity everywhere) as λ approaches infinity (which is the appropriate value for the incompressible problem). One practical way around this difficulty is to use inexact numerical quadrature when evaluating the integrals. It is recommended by Hughes, *et al.* [1979] that we use the quadrature as shown in Figure 12.1. This figure shows

the types of trial functions and the number of Gauss points that should be used for terms multipled by μ and λ. The Newton-Raphson method is also applied to Eq. (12.15) to obtain the equations to be solved iteratively. This completes the formulation of the penalty method for the Navier-Stokes equation. Comparisons of this method with the artificial compressibility method are available [Shih, *et al.*, 1989].

The primitive (u-p-v) method for the Navier-Stokes equation for an incompressible fluid is unusual in that the equation for pressure (the continuity equation) does not have the pressure variable in it. The continuity equation acts like a constraint on the velocity field, leading to the restriction that the pressure and velocities must be expanded in different trial functions. The stream function-vorticity method avoids this difficulty because the incompressibility constraint is satisfied automatically. This method introduces another problem, however, because on a solid surface there are two boundary conditions for stream function and none for vorticity. This presents little difficulty for a finite element method. When solving these equations, it is highly recommended that we solve them together as a coupled system since the boundary conditions are not provided for vorticity on all surfaces. It is possible to separate the problems, as is done with finite difference methods, but experience suggests it is better not to do so (Campion-Renson and Crochet [1978] and Stevens [1982]).

12.3. Finite Difference Methods

There are a variety of finite difference methods for the Navier-Stokes equation. Usually the various methods are designed to operate in an iterative fashion, even for a zero Reynolds number. This is advantageous because large computer memories are not necessary since no matrices are "inverted." However, there are also disadvantages, as indicated below for each finite difference method. In contrast, in finite element methods we obtain a large matrix problem to be solved at each iteration; for $Re = 0$ only one iteration is needed.

The first finite difference method, which originated with Chorin [1967], allows the fluid to have a small value of compressibility. The chief problem with the Navier-Stokes equation and the continuity equation is that the Navier-Stokes equation is an equation for velocity and it includes the pressure gradient, while the continuity equation is a constraint on the velocity but is in essence the equation for pressure. In the artificial compressibility method a time derivative of pressure is added to the continuity equation, as shown in Table 12.2. The constant parameter c^2 is chosen to aid in convergence; the transient itself does not have any physical significance. This method is called the artificial compressibility method because the pressure equation is the continuity equation for a compressible fluid with an equation of state

$$p = c^2 \rho. \tag{12.16}$$

(See the electronic text entitled "Compressibility Method.")

Table 12.2. Finite Difference Methods for the Navier-Stokes Equations

Artificial Compressibility

$$\frac{\partial \mathbf{u}}{\partial t} = -\mathbf{u} \cdot \nabla \mathbf{u} - \nabla p + \frac{1}{Re} \nabla^2 \mathbf{u}$$

$$\frac{\partial p}{\partial t} = -c^2 \nabla \cdot \mathbf{u}$$

Unsteady-State

$$\frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = -\mathbf{u}^n \cdot \nabla \mathbf{u}^n + \frac{1}{Re} \nabla^2 \mathbf{u}^n$$

$$\nabla^2 p^{n+1} = \frac{1}{\Delta t} \nabla \cdot \mathbf{u}^*$$

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}^*}{\Delta t} = -\nabla p^{n+1}$$

Stream Function-Vorticity

$$\mathbf{u} \cdot \nabla \xi - \frac{1}{Re} \nabla^2 \xi = 0, \quad \nabla^2 \psi = -\xi$$

$$\tau \cdot \nabla \psi = \mathbf{u}_s \cdot \mathbf{n}, \quad \mathbf{n} \cdot \nabla \psi = -\mathbf{u}_s \cdot \tau$$

The second finite difference method is a pseudo-steady-state method that splits the Navier-Stokes equation into two parts, one involving the velocity terms and one involving the pressure terms (see Table 12.2). After computing $\mathbf{u}^*$ using the first equation, we solve a Poisson equation for pressure. This equation is derived by taking the divergence of the third equation and setting the divergence of $\mathbf{u}^{n+1}$ equal to zero. After solving the Poisson equation for pressure, we solve for $\mathbf{u}^{n+1}$ using the third equation. The calculations are straightforward for the velocity terms. For the pressure terms we must solve the Poisson equation. This can be done using iterative techniques, such as Gauss-Seidel or overrelaxation methods. One boundary condition that can be used with this Poisson equation is a zero flux condition (Peyret and Taylor [p. 162, 1983]), provided that the velocity $\mathbf{u}^*$ satisfies the same boundary conditions as $\mathbf{u}^{n+1}$. The actual boundary condition is obtained by taking the normal component of the third equation but the value of $\mathbf{u}^*$ drops out of the Poisson equation. What is being solved for in the pseudo-steady-state method is not pressure but velocity, and this velocity is the correct only at steady-state. The correct pressure can be obtained by taking the divergence of the Navier-Stokes equation to give

$$\nabla^2 p = -\nabla \cdot (\mathbf{u} \cdot \nabla \mathbf{u}), \tag{12.17}$$

$$\mathbf{n} \cdot \nabla p = -\mathbf{n} \cdot (\mathbf{u} \cdot \nabla \mathbf{u}) + \frac{1}{Re} \mathbf{n} \cdot \nabla^2 \mathbf{u}. \tag{12.18}$$

The divergence of the Laplacian of the velocity is the same as the Laplacian of the divergence of the velocity, which is zero according to the continuity equation. The finite difference equations are therefore

$$\frac{u^*_{i+1/2,j} - u^n_{i+1/2,j}}{\Delta t} = -u^n_{i+1/2,j}\frac{u^n_{i+3/2,j} - u^n_{i-1/2,j}}{2\Delta x} +$$
$$-\frac{1}{4}(v^n_{i+1,j+1/2} + v^n_{i,j+1/2} + v^n_{i,j-1/2} + v^n_{i+1,j-1/2})\frac{u^n_{i+1/2,j+1} - u^n_{i+1/2,j-1}}{2\Delta y} +$$
$$+\frac{1}{Re}\left(\frac{u^n_{i+3/2,j} - 2u^n_{i+1/2,j} + u^n_{i-1/2,j}}{\Delta x^2} + \frac{u^n_{i+1/2,j+1} - 2u^n_{i+1/2,j} + u^n_{i+1/2,j-1}}{\Delta y^2}\right), \tag{12.19}$$

$$\frac{v^*_{i,j+1/2} - v^n_{i,j+1/2}}{\Delta t} = -v^n_{i,j+1/2}\left(\frac{v^n_{i,j+3/2} - v^n_{i,j-1/2}}{2\Delta y}\right)$$
$$-\frac{1}{4}(u^n_{i+1/2,j} + u^n_{i+1/2,j+1} + u^n_{i-1/2,j} + u^n_{i-1/2,j+1})\frac{v^n_{i+1,j+1/2} - v^n_{i-1,j+1/2}}{2\Delta x} +$$
$$+\frac{1}{Re}\left(\frac{v^n_{i+1,j+1/2} - 2v^n_{i,j+1/2} + v^n_{i-1,j+1/2}}{\Delta x^2} + \frac{v^n_{i,j+3/2} - v^n_{i,j+1/2} + v^n_{i,j-1/2}}{\Delta y^2}\right), \tag{12.20}$$

$$\frac{p^{n+1}_{i+1,j} - 2p^{n+1}_{i,j} + p^{n+1}_{i-1,j}}{\Delta x^2} + \frac{p^{n+1}_{i,j+1} - 2p^{n+1}_{i,j} + p^{n+1}_{i,j-1}}{\Delta y^2} =$$
$$= -\frac{1}{\Delta t}\left[\frac{u^*_{i+1/2,j} - u^*_{i-1/2,j}}{\Delta x} + \frac{v^*_{i,j+1/2} - v^*_{i,j-1/2}}{\Delta y}\right] \tag{12.21}$$

$$\frac{u^{n+1}_{i+1/2,j} - u^*_{i+1/2,j}}{\Delta t} = -\frac{p^{n+1}_{i+1,j} - p^{n+1}_{i,j}}{\Delta x} \tag{12.22}$$

$$\frac{v^{n+1}_{i,j+1/2} - v^*_{i,j+1/2}}{\Delta t} = -\frac{p^{n+1}_{i,j+1} - p^{n+1}_{i,j}}{\Delta y}. \tag{12.23}$$

Finite difference methods can also be applied to the stream function-vorticity equations, which are usually solved iteratively.

12.4. Finite Element Methods for Unsteady-State Problems

Finite element methods are usually applied to steady-state problems by solving the set of algebraic equations [Eq. (12.11)-(12.12)] using the Newton-Raphson method. Finite difference equations are usually solved using relaxation or other iterative methods, or a pseudo-unsteady-state method is used. New finite element methods must be developed for the transient case. The first method discussed below has been developed by Gresho and co-workers [1984]. It is a classical finite element method, although several approximations are made to

improve the efficiency. The second method employs the method of characteristics to move material points and is in fact similar to some of the finite difference methods. Three other finite element methods are briefly mentioned at the end of this section. The next section incorporates the ideas of Taylor-Galerkin methods.

Modified finite element method. For transient problems, we expand the velocity and pressures as in Eq. (12.1), except that now the coefficients are functions of time rather than constants. When these expansions are inserted into the Navier-Stokes equation and the continuity equation, we obtain the following set of differential-algebraic equations:

$$M\dot{u} + N(u)\,u + K\,u + C\,P = f, \tag{12.24}$$

$$C^T u = 0. \tag{12.25}$$

The first term of equation (12.24) represents the mass matrix, which appears in the finite element treatment of time derivatives. The second term represents the inertial terms, the third term is the viscous term, and the remaining terms come from the pressure gradient and body forces. The second equation is a discrete representation of the continuity equation. These equations are inconvenient to solve because they have time derivatives for the velocities in the two velocity equations, but no time derivative for pressure in the continuity equation. Thus we derive a discrete Poisson equation for pressure in the manner used to derive Eq. (12.17). We write the discrete Navier-Stokes equation as

$$C\,P + M\dot{u} = f - N(u)\,u - K\,u. \tag{12.26}$$

We multiply this by M^{-1}, giving

$$M^{-1}CP + \dot{u} = M^{-1}\,[\,f - N(u)\,u - K\,u]. \tag{12.27}$$

Next we multiply Eq. (12.27) by C^T, giving

$$C^T M^{-1} CP + C^T\dot{u} = C^T M^{-1}\,[\,f - N(u)\,u - K\,u] \tag{12.28}$$

and differentiate the continuity equation with respect to time. Thus

$$C^T\dot{u} = 0 \tag{12.29}$$

and the equation for pressure becomes

$$C^T M^{-1} CP = C^T M^{-1}\,[\,f - N(u)\,u - K\,u\,]. \tag{12.30}$$

This is a discrete approximation of Eq. (12.17). To solve this equation we must apply boundary conditions on the pressure, which should be the Neumann conditions given in Eq. (12.18) [Gresho and Sani, 1987].

In order to solve Eq. (12.24) using Eq. (12.30), we need an efficient way to calculate M^{-1} as well as to solve Eq. (12.30). The first method is an explicit

method. Gresho, *et al.* [1984] use a lumped mass matrix so that M is diagonal and its inverse is easy to find. Then, for further efficiency, they evaluate the coefficient matrices using one-point quadrature; an hourglass correction is needed for the diffusion terms to improve the results. Finally, they use the Euler method to integrate Eq. (12.24) in time, with the pressure equation [Eq. (12.30)] solved only occasionally. We note that the matrix problem in Eq. (12.30) need be solved only once per problem since it does not change from time-step to time-step. Even so, the matrix problem is a time-consuming and expensive calculation; the pressure does not change rapidly, so it is updated on a longer time-step than the velocity. In order to do this, it is necessary for the initial approximation of velocity to be solenoidal,

$$C^T u_0 = 0, \tag{12.31}$$

so that the initial transients arising from Eq. (12.29) are not severe.

The algorithm is sufficiently important to be written in detail. First let us look at the continuity equation, Eq. (12.25). We suppose that we have N+M number of velocity unknowns, with M of them specified on boundaries. We separate the vector of velocity nodal values into two vectors, one of length N and the other of length M. The continuity equation can then be written in the form of Figure 12.2. The first N equations can be written as

$$C_{11}^T u = - C_{12}^T u_B, \tag{12.32}$$

where the right-hand side is known. We call it g(t). Then Eq. (12.32) becomes

$$C_{11}^T u = g(t) \tag{12.33}$$

and Eq. (12.29) becomes

$$C_{11}^T \dot{u} = \frac{dg}{dt}. \tag{12.34}$$

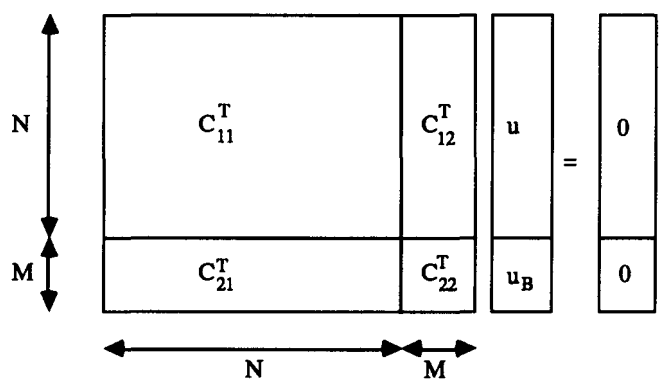

Figure 12.2. Matrix Form for the Continuity Equation

We start with a velocity that satisfies the essential boundary conditions (specified velocities) and Eq. (12.33) at time zero. Then we solve for pressure using the equation

$$C^T M^{-1} C P^n = C^T M^{-1} [\, f^n - N(u^n)\, u^n - K\, u^n \,] - \frac{g^{n+1} - g^n}{\Delta t}. \tag{12.35}$$

All terms on the right-hand side are known since g^{n+1} is known from the boundary conditions. Eq. (12.35) is derived from Eq. (12.28) using Eq. (12.34). It is the equivalent of Eq. (12.30) but with the boundary conditions taken into account. There are two types of boundary conditions. On the part of the boundary where the velocity is specified, the boundary conditions are the Neumann conditions, derived as the normal component of Eq. (12.24),

$$\frac{\partial p}{\partial n} = \mathbf{n} \cdot \left(\mathbf{f} + \mu \nabla^2 \mathbf{u} - \rho \frac{\partial \mathbf{u}}{\partial t} - \rho\, \mathbf{u} \cdot \nabla\ \mathbf{u} \right). \tag{12.36}$$

On the rest of the boundary, the pressure is calculated from Eq. (12.4).

$$p = \mu\, \mathbf{n} \cdot [\, \nabla\ \mathbf{u} + \nabla\ \mathbf{u}^T \,] - f_n \tag{12.37}$$

Finally, the velocity is updated to give

$$M \frac{u^{n+1} - u^n}{\Delta t} = - C P^n + f^n - N(u^n)\, u^n - K\, u^n. \tag{12.38}$$

For efficiency in this version, the mass matrix is lumped in all occurrences. The final algorithm is then (where M_L represents a lumped mass matrix)

$$C^T M_L^{-1} C P^n = C^T M_L^{-1} [\, f^n - N(u^n)\, u^n - K\, u^n \,] - \frac{g^{n+1} - g^n}{\Delta t}, \tag{12.39}$$

$$M_L \frac{u^{n+1} - u^n}{\Delta t} = - C P^n + f^n - N(u^n)\, u^n - K\, u^n. \tag{12.40}$$

This algorithm is a finite element version of the finite difference method discussed in Section 12.3, based on the unsteady-state version listed in Table 12.2. If a finite element method were applied directly to the unsteady-state equations as listed in Table 12.2, we would get the algorithm

$$M \frac{\tilde{u}^{n+1} - u^n}{\Delta t} = f^n - N(u^n)\, u^n - K\, u^n, \tag{12.41}$$

$$C^T M^{-1} C P^{n+1} = \frac{1}{\Delta t} C^T \tilde{u}^{n+1}, \tag{12.42}$$

$$M \frac{u^{n+1} - \tilde{u}^{n+1}}{\Delta t} = - C P^{n+1}. \tag{12.43}$$

If we add Eq. (12.41) to Eq. (12.43), we get the momentum equation:

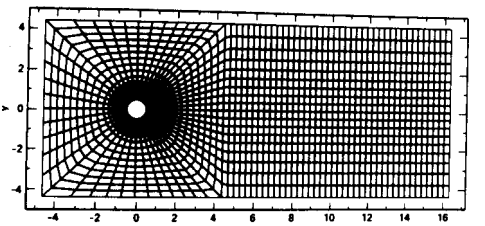

Figure 12.3a. Finite Element Mesh for Flow Past a Cylinder

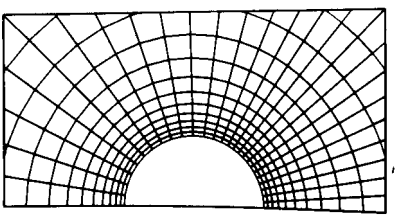

Figure 12.3b. Enlargement of the Finite Element Mesh for Flow Past a Cylinder
From Gresho, *et al.*, [1984]; copyright 1984; reprinted by permission of John Wiley & Sons, Ltd.

$$M \frac{u^{n+1} - u^n}{\Delta t} = - C P^{n+1} + f^n - N(u^n) u^n - K u^n. \qquad (12.44)$$

How are the finite element equations, Eq. (12.41)-(12.44), related to the modified finite element method as summarized in Eq. (12.35)-(12.38)? To see the relation, we take a case where the velocity boundary conditions are constant in time; then $g^{n+1} = g^n$ in Eq. (12.35). We define a new variable, $\tilde{u}^{n+1}$, such that

$$f^n - N(u^n) u^n - K u^n = M \frac{\tilde{u}^{n+1} - u^n}{\Delta t}. \qquad (12.45)$$

Then Eq. (12.35) is written as

$$C^T M^{-1} C P^n = C^T M^{-1} M \frac{\tilde{u}^{n+1} - u^n}{\Delta t} = C^T \frac{\tilde{u}^{n+1} - u^n}{\Delta t}. \qquad (12.46)$$

At the initial time, the equation $C^T u^n = 0$ holds; this equation is then forced to be true for subsequent times. Eq. (12.46) is equivalent to Eq. (12.42), except that the result is called P^{n+1} instead of P^n. Thus the final equations, Eq. (12.38) and Eq. (12.44), are in reality the same; the method used by Gresho and co-workers [Gresho and Chan, 1985] is the finite element analog to the finite difference method listed as "Unsteady-State" in Table 12.2.

Gresho and co-workers [Gresho and Chan, 1985] have developed their method further to allow larger time-steps. The algorithm represented by Eq.

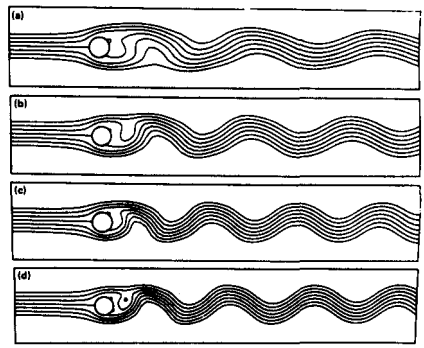

Figure 12.4. Streamlines During Vortex Shedding; Re = (a) 50, (b) 100, (c) 200, (d) 400. From Gresho, *et al.*, [1984]; copyright 1984; reprinted by permission of John Wiley & Sons, Ltd.

(12.38) has a step-size that is limited by diffusion, i.e., $\Delta t \leq c\,\mu\,\Delta x^2$ for some value of c. When the viscosity is small but non-zero, this limitation requires a very small step-size. To improve on this, the semi-implicit method was adopted. We merely replace the diffusion terms in Eq. (12.40) with an implicit version:

$$C^T M_L^{-1} C P^n = C^T M_L^{-1} [\, f^n - N(u^n)\, u^n - K\, u^n \,] - \frac{g^{n+1} - g^n}{\Delta t}, \tag{12.47}$$

$$M_L \frac{u^{n+1} - u^n}{\Delta t} + K\, u^{n+1} = - C P^n + f^n - N(u^n)\, u^n. \tag{12.48}$$

The method can be improved even more if the mass matrix is not lumped in the momentum equation. The pressure equation is easily and quickly solved when the mass matrix is lumped, so we keep that approximation in the pressure equation. Then Eq. (12.47)-(12.48) are replaced by

$$C^T M_L^{-1} C P^n = C^T M_L^{-1} [\, f^n - N(u^n)\, u^n - K\, u^n \,] - \frac{g^{n+1} - g^n}{\Delta t}, \tag{12.49}$$

$$M \frac{u^{n+1} - u^n}{\Delta t} + K\, u^{n+1} = - C P^n + f^n - N(u^n)\, u^n. \tag{12.50}$$

An additional viscosity term is added that is proportional to $\Delta t u_i u_j/2$; it is called a balancing tensor diffusivity. The reason for using this term is closely connected to the Taylor-Galerkin method, which is outlined in Section 12.5. Eq. (12.49)-(12.50) gives good results, too [Gresho and Chan, 1988].

The explicit method [Eq. (12.47)-(12.48)] has been applied to the problem of flow past a cylinder at large Reynolds numbers. The mesh is as shown in Figure 12.3 [Gresho, *et al.*, 1984]. For Reynolds numbers above 50, the steady-state solution to this problem is unstable and an unsteady flow develops. Vortices are shed behind the cylinder one after another with a frequency f. A typical view

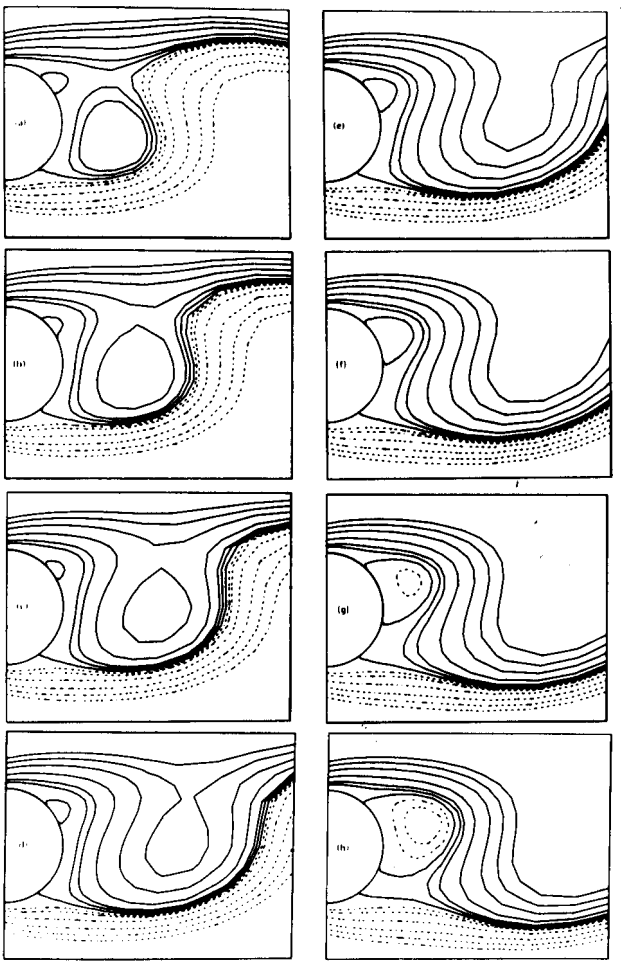

Figure 12.5. Streamlines Near Cylinder, Re = 50, $\Delta t = \tau/16$ for Successive Pictures From Gresho, *et al.*, [1984]; copyright 1984; reprinted by permission of John Wiley & Sons, Ltd.

of the streamlines behind the cylinder is shown in Figure 12.4. For one Reynolds number, the successive streamlines are shown as a function of time in Figure 12.5. The frequency at which the vortices are shed is a convenient measure for comparing experiments; this dimensionless frequency is called the Strouhal number,

$$St = \frac{u_0 f}{D}, \tag{12.51}$$

where u_0 is the upstream velocity of the fluid past a cylinder with diameter D. This agrees reasonably well with experimental correlations (0.14 versus 0.12 for a Reynolds number = 50). The calculations using the first algorithm discussed,

Eq. (12.24)-(12.40), took about two hours on a CRAY computer. See also Kovacs and Kawahara [1991] for additional applications.

Donea, *et al.* [1982] presented a fractional step finite element method in which the momentum equation was split into two steps. The convective and diffusive terms were used to calculate the velocity at the n+1/2-th time-step and a Poisson equation for pressure was used to obtain the pressure at the n+1-th step. Then the velocity at the n+1-th step was obtained, including the pressure term. Similar ideas for the Taylor-Galerkin method are described in detail in Section 12.5.

Method of characteristics with operator splitting. The second method is one developed by Huffenus and Khaletzky [1984] that involves operator splitting. They employ a method of characteristics to follow the movement of fluid elements. If we solve the hyperbolic equation

$$\frac{\partial F}{\partial t} + u\frac{\partial F}{\partial x} = 0 \tag{12.52}$$

to move from time level n to time level n+1, then the solution at node i and time level n+1 is the value of the function at F*, as illustrated in Figure 12.6.

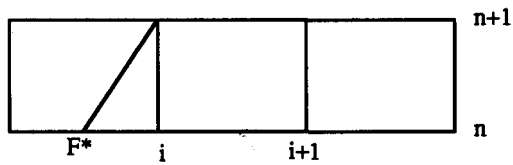

Figure 12.6. Method of Characteristics to Go from n-th Time level to n+1-th Time Level

Thus, when solving

$$\frac{dM}{dt} = V(M,t), \tag{12.53}$$

we use

$$M^* = M_i^{n+1} - V\Delta t\,, \tag{12.54}$$

where the velocity is constant during the time interval from time level n to time level n+1. The method works as follows. We use the method of characteristics to obtain u* and v* at each node (i.e., the velocity of the fluid at node i and time level n+1). Then we solve

$$\nabla^2 p = \frac{\nabla \cdot \mathbf{v}^*}{\Delta t} \tag{12.55}$$

using the Neumann boundary conditions from the momentum equation. Finally, we solve

$$\frac{\mathbf{v}^{n+1} - \mathbf{v}^*}{\Delta t} = -\nabla\, p^{n+1} + \frac{1}{Re}\nabla^2 \mathbf{v}^{n+1} \tag{12.56}$$

to determine the velocity at the new time level. The steps of the method are similar

to those used in the unsteady-state finite difference method listed in Table 12.2, except that the convection terms are handled using the method of characteristics, Eq. (12.54), rather than in the standard way. Huffenus and Khaletzky [1984] use triangular elements with quadratic velocities and linear pressures to solve the driven cavity problem. Yang, et al. [1991] use a similar method for the convective diffusion equation.

Compressible-fluid method. The third method, based on work by Kawahara [1983], allows for density to depend on pressure so that the fluid is effectively compressible (although the compressibility may be small). In this way, the continuity equation is changed from a differential equation constraint to a time-dependent equation for pressure. After the compressible fluid finite element method is applied in space, the unsteady-state problem has three ordinary differential equations for u, v, and p, which can be solved using a variety of methods. Kawahara [1983] uses an explicit method; he also solves the problem of vortex shedding behind a cylinder.

Stream function-vorticity method. The fourth method is a time-dependent application of the stream function-vorticity method by Miyauchi, *et al.* [1983]. In this case, the time-dependent vorticity equation is integrated using either the Euler method or a Lax-Wendroff method in time.

Upstream method. The final method comes from work by Hughes, *et al.* [1979]: a penalty method is used along with upstream evaluation of the convection matrices. Lobatto quadrature is used, which in the lowest order is the trapezoid rule and in the next highest order is Simpson's rule. This has the effect of lumping the mass. When the steady-state equations are written as

$$\mathbf{C}\,\mathbf{v} + \mathbf{N}(\mathbf{v}) = \mathbf{F}, \tag{12.57}$$

then the transient method is as follows. From the n-th time level we know the value of $\mathbf{v}_n$ and $\mathbf{a}_n$. The algorithm is then

$$\begin{aligned}
&\mathbf{v}_{n+1} = \mathbf{v}_n + (1-\gamma)\,\mathbf{a}_n,\\
&\mathbf{v}^0_{n+1} = \tilde{\mathbf{v}}_{n+1},\\
&(\mathbf{M} + \gamma\,\Delta t\,\mathbf{C})\,\mathbf{v}^{k+1}_{n+1} = \mathbf{M}\,\tilde{\mathbf{v}}_{n+1} + \gamma\,\Delta t\,[\,\mathbf{F} - \mathbf{N}\,(\mathbf{v}^k_{n+1})],\\
&\mathbf{a}_{n+1} = (\,\mathbf{v}_{n+1} - \tilde{\mathbf{v}}_{n+1}\,)/\gamma\,\Delta t\,.
\end{aligned} \tag{12.58}$$

Apparently the equation is iterated until it converges (at each time-step). The parameter γ is set as $\geq 1/2$ for stability reasons. The penalty parameter takes values similar to the parameter for a steady-state simulation. Since upstream effects are included, the Reynolds number can be very large. One solution is given for flow past a step in the boundary; starting from a steady-state solution for a Reynolds number of 30, the flow rate is increased to an effective Reynolds number of 10^7. No attempt is made to determine the influence of upstream weighting on the type of solution, but the solution is "reasonable-looking," even

if there is so much upstream diffusion that it is incorrect. Since real flows usually become turbulent at such large Reynolds numbers, determining whether or not it is an appropriate solution of the equations must await more detailed studies with finer meshes.

For unsteady flows it is also possible to move the mesh in some way. Ramaswamy and Kawahara [1987] use a combined Lagrangian-Eulerian finite element method for treating time-dependent flows with free surfaces. Gopalakrishnan [1988] moved the mesh using local kinematics.

The **flux-correction method** has also been applied to compressible flow equations and the Navier-Stokes equation [Löhner, *et al.*, 1987 and Thareja, *et al.*, 1989].

12.5. The Taylor-Galerkin Method

The Taylor-Galerkin method can be applied to the unsteady-state Navier-Stokes equation,

$$\rho\left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla\ \mathbf{u}\right) = \rho\,\mathbf{f} - \nabla\ p + \nabla \cdot [\mu(\nabla\ \mathbf{u} + \nabla\ \mathbf{u}^T)], \tag{12.59}$$

which must be solved together with the continuity equation,

$$\nabla \cdot \mathbf{u} = 0. \tag{12.60}$$

As before, we expand the velocity in a Taylor series in time giving

$$\mathbf{u}^{n+1} = \mathbf{u}^n + \left.\frac{\partial \mathbf{u}}{\partial t}\right|^n \Delta t + \left.\frac{\partial^2 \mathbf{u}}{\partial t^2}\right|^n \frac{\Delta t^2}{2}. \tag{12.61}$$

This equation is rearranged to give

$$\left.\frac{\partial \mathbf{u}}{\partial t}\right|^n = \frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{\Delta t} - \frac{\Delta t}{2}\left.\frac{\partial^2 \mathbf{u}}{\partial t^2}\right|^n. \tag{12.62}$$

We evaluate the second time derivative from the unsteady-state Navier-Stokes equation by using Cartesian tensor notation and transforming the final result into vector notation to make it general. Let us consider only the convective term in the Navier-Stokes equation:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = 0 \tag{12.63}$$

or

$$\frac{\partial u_i}{\partial t} = -u_j \frac{\partial u_i}{\partial x_j}. \tag{12.64}$$

We can also write this in conservative form as

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}(u_j u_i) = 0 \tag{12.65}$$

or

$$\frac{\partial u_i}{\partial t} = -\frac{\partial}{\partial x_j}(u_j u_i). \tag{12.66}$$

We differentiate Eq. (12.66) once with respect to time, giving

$$\frac{\partial^2 u_i}{\partial t^2} = -\frac{\partial}{\partial x_j}\left(u_j \frac{\partial u_i}{\partial t}\right) - \frac{\partial}{\partial x_j}\left(\frac{\partial u_j}{\partial t} u_i\right). \tag{12.67}$$

Then we insert the non-conservative form, Eq. (12.64), which gives

$$\frac{\partial^2 u_i}{\partial t^2} = \frac{\partial}{\partial x_j}\left(u_j u_k \frac{\partial u_i}{\partial x_k}\right) + \frac{\partial}{\partial x_j}\left(u_k \frac{\partial u_j}{\partial x_k} u_i\right). \tag{12.68}$$

Finally, we write this in vector notation:

$$\frac{\partial^2 \mathbf{u}}{\partial t^2} = \nabla \cdot [\mathbf{u}(\mathbf{u} \cdot \nabla \mathbf{u})] + \nabla \cdot [(\mathbf{u} \cdot \nabla \mathbf{u})\mathbf{u}]. \tag{12.69}$$

The unsteady-state Navier-Stokes equation is then

$$\rho\left(\frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{\Delta t} + \mathbf{u} \cdot \nabla \mathbf{u}\right) = \rho \mathbf{f} - \nabla p + \nabla \cdot [\mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)] + \frac{\rho \Delta t}{2} \frac{\partial^2 \mathbf{u}}{\partial t^2}\Bigg|^n, \tag{12.70}$$

which becomes

$$\rho\left(\frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{\Delta t} + \mathbf{u} \cdot \nabla \mathbf{u}\right) = \rho \mathbf{f} - \nabla p + \nabla \cdot [\mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)] + \\ + \frac{\rho \Delta t}{2}\{\nabla \cdot [\mathbf{u}(\mathbf{u} \cdot \nabla \mathbf{u})] + \nabla \cdot [(\mathbf{u} \cdot \nabla \mathbf{u})\mathbf{u}]\}. \tag{12.71}$$

For us to see the effect of the additional terms, we should write the equations in component notation. We use cylindrical geometry and take a case with azimuthal symmetry. We use z for the axial coordinate and r for the radial coordinate and also use the condition

$$u_\theta = 0 \tag{12.72}$$

and allow no θ-variation in the solution. The velocities are

$$\mathbf{e}_z \cdot \mathbf{u} = u \text{ and } \mathbf{e}_r \cdot \mathbf{u} = v. \tag{12.73}$$

The convective terms are given on the next page.

$$\begin{aligned} &\text{The } \mathbf{e}_z \text{ component of } \mathbf{u}\cdot\nabla\ \mathbf{u} \text{ is } v\frac{\partial u}{\partial r}+u\frac{\partial u}{\partial z}.\\ &\text{The } \mathbf{e}_r \text{ component of } \mathbf{u}\cdot\nabla\ \mathbf{u} \text{ is } v\frac{\partial v}{\partial r}+u\frac{\partial v}{\partial z}. \end{aligned} \tag{12.74}$$

The Taylor terms involve the dyadic $\mathbf{u}(\mathbf{u}\cdot\nabla\ \mathbf{u})+(\mathbf{u}\cdot\nabla\ \mathbf{u})\mathbf{u}$. Its components are

$$\begin{aligned} &\mathbf{e}_z\mathbf{e}_z && 2\left(uv\frac{\partial u}{\partial r}+u^2\frac{\partial u}{\partial z}\right),\\ &\mathbf{e}_z\mathbf{e}_r \text{ and } \mathbf{e}_r\mathbf{e}_z && u\left(v\frac{\partial v}{\partial r}+u\frac{\partial v}{\partial z}\right)+\left(v\frac{\partial u}{\partial r}+u\frac{\partial u}{\partial z}\right)v, \end{aligned} \tag{12.75}$$

$$\begin{aligned} &\mathbf{e}_r\mathbf{e}_r && 2\left(v^2\frac{\partial v}{\partial r}+uv\frac{\partial v}{\partial z}\right),\\ &\mathbf{e}_\theta\mathbf{e}_\theta && 0. \end{aligned} \tag{12.76}$$

We note that the dyadic is symmetric. Now a second-order dyadic can be written in terms of its components (in this geometry) as

$$\mathbf{A}=\mathbf{e}_z\mathbf{e}_z A_{zz}+\mathbf{e}_z\mathbf{e}_r A_{zr}+\mathbf{e}_r\mathbf{e}_z A_{rz}+\mathbf{e}_r\mathbf{e}_r A_{rr}+\mathbf{e}_\theta\mathbf{e}_\theta A_{\theta\theta}. \tag{12.77}$$

For the divergence, the components of $\nabla\cdot\mathbf{A}$ are

$$\begin{aligned} &\mathbf{e}_z && \frac{1}{r}\frac{\partial}{\partial r}(rA_{rz})+\frac{\partial}{\partial z}(A_{zz}),\\ &\mathbf{e}_r && \frac{1}{r}\frac{\partial}{\partial r}(rA_{rr})+\frac{\partial}{\partial z}(A_{zr})-\frac{A_{\theta\theta}}{r}. \end{aligned} \tag{12.78}$$

For the Taylor terms, the components of $\nabla\cdot[\mathbf{u}(\mathbf{u}\cdot\nabla\mathbf{u})+(\mathbf{u}\cdot\nabla\mathbf{u})\mathbf{u}]$ are

$$\begin{aligned} &\mathbf{e}_z && \frac{1}{r}\frac{\partial}{\partial r}\left\{r\left[v^2\frac{\partial u}{\partial r}+uv\frac{\partial u}{\partial z}+vu\frac{\partial v}{\partial r}+u^2\frac{\partial v}{\partial z}\right]\right\}+\frac{\partial}{\partial z}\left\{2uv\frac{\partial u}{\partial r}+2u^2\frac{\partial u}{\partial z}\right\},\\ &\mathbf{e}_r && \frac{1}{r}\frac{\partial}{\partial r}\left\{r\left[2v^2\frac{\partial v}{\partial r}+2uv\frac{\partial v}{\partial z}\right]\right\}+\frac{\partial}{\partial z}\left\{uv\frac{\partial v}{\partial r}+u^2\frac{\partial v}{\partial z}+v^2\frac{\partial u}{\partial r}+uv\frac{\partial u}{\partial z}\right\}. \end{aligned} \tag{12.79}$$

Let us next consider the case where the dyadic $\mathbf{A}$ represents a shear stress for a Newtonian fluid. The components are

$$\tau_{zz}=-2\mu\frac{\partial u}{\partial z},\quad \tau_{rz}=\tau_{zr}=-\mu\left(\frac{\partial u}{\partial r}+\frac{\partial v}{\partial z}\right),\quad \tau_{rr}=-2\mu\frac{\partial v}{\partial r},\quad \tau_{\theta\theta}=-2\mu\frac{v}{r}. \tag{12.80}$$

By comparing Eq. (12.75)-(12.76) with Eq. (12.80) we can see that the Taylor terms add viscous-like terms to the equations. In fact, we can write Eq. (12.75)-(12.76) as

$$\mathbf{A}=\mathbf{T}:\nabla\ \mathbf{u} \tag{12.81}$$

and deduce the fourth-order dyadic $\mathbf{T}$. The Taylor term can then be regarded as an anisotropic viscous term.

An alternative formulation of the Taylor-Galerkin method for the Navier-Stokes equation was given by Laval [1988]. She expresses the method in terms of operator splitting. First there is a convection step,

$$\rho\left(\frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} + \mathbf{u}\cdot\nabla\ \mathbf{u}\right) = \frac{\rho\,\Delta t}{2}\{\nabla\cdot[\mathbf{u}^n(\mathbf{u}^n\cdot\nabla\ \mathbf{u}^n)] + \nabla\cdot[(\mathbf{u}^n\cdot\nabla\ \mathbf{u}^n)\mathbf{u}^n]\}, \tag{12..82}$$

$$\mathbf{u}^* = \mathbf{b}^{n+1} \text{ on inflow boundaries,} \tag{12..83}$$

where **b** represents the inlet velocity. Then there is a viscous step,

$$\rho\,\frac{\mathbf{u}^{**} - \mathbf{u}^*}{\Delta t} = \rho\,\mathbf{f} + \nabla\cdot[\mu(\nabla\ \mathbf{u}^* + \nabla\ \mathbf{u}^{*T})], \tag{12.84}$$

$$\mathbf{u}^{**} = \mathbf{b}^{n+1} \text{ on boundaries with velocity specified.} \tag{12.85}$$

The pressure is obtained by solving

$$\rho\,\frac{\nabla\cdot\mathbf{u}^{**}}{\Delta t} = -\nabla^2 p^{n+1}. \tag{12.86}$$

Finally, $\mathbf{u}^{n+1}$ can be calculated:

$$\rho\,\frac{\mathbf{u}^{n+1} - \mathbf{u}^{**}}{\Delta t} = -\nabla\ p^{n+1}, \text{ and } \nabla\cdot\mathbf{u}^{n+1} = 0, \tag{12.87}$$

$$\mathbf{u}^{n+1}\cdot\mathbf{n} = \mathbf{b}^{n+1}\cdot\mathbf{n} \text{ on boundaries where velocity is specified.} \tag{12.88}$$

Comparison of Eq. (12.82)-(12.88) with Eq. (12.41)-(12.43) shows that they are the same, except that the convection and viscous step are separated into two steps and the diffusion is evaluated at the predicted velocity **u***. This means that the method is similar to Gresho's formulation when the same finite element decisions are made (i.e., the same trial functions, decisions on lumping, etc.). In the two-step version, Eq. (12.82)-(12.88), methods optimized for each step could be used. Other examples of similar methods are by Ramaswamy [1990], Hawken, *et al.* [1990], and Pepper and Singer [1990].

For the compressible high speed flow equations, Löhner, *et al.* [1985] used a two-step formulation of the Taylor-Galerkin method that is similar to the MacCormack and Lax-Wendroff methods. The two-step formulation proved faster than the one-step formulation because the Jacobian was less complicated. The compressible flow equations were without viscosity but artificial viscosity was added proportional to the velocity gradients. Löhner, *et al.* also did adaptive remeshing by using a criterion that the local error should be constant. The local error was given by

$$\|u - u^h\|_0^e < C\,h_e^2\ |u|_2^e. \tag{12.89}$$

Thus they chose the mesh size so that

$$h_e^2 \, |u|_2^e = \text{constant.} \tag{12.90}$$

The norm was defined as

$$|u|_2 = \sqrt{\int_\Omega \left[\left(\frac{\partial^2 u}{\partial x^2} \right)^2 + 2 \left(\frac{\partial^2 u}{\partial x \partial y} \right)^2 + \left(\frac{\partial^2 u}{\partial y^2} \right)^2 \right] d\Omega} \tag{12.91}$$

and Löhner, *et al.* [1985] used a finite element approximation to this norm. (See also Löhner [1987]). Oden, *et al.* [1986] treated the same equations, using artificial viscosity and adaptive mesh refinement. For a group of four elements, the center node could be moved or the elements could be subdivided to achieve an equidistribution criterion based on the estimated error.

References

Campion-Renson, A. and Crochet, M. J., "On the Stream Function-Vorticity Finite Element Solutions of Navier-Stokes Equations, Int. J. Num. Methods Engn. 12 1809-1818 [1978].

Chorin, A. J., "A Numerical Method for Solving Incompressible Viscous Flow Problems," J.Comp.Phys.2 12-26 [1967].

Cuvelier, C., Segal, A. and van Steenhoven, A. A., *Finite Element Methods and Navier-Stokes Equations*, Reidel Publishing, Dordrecht [1986].

Donea, J., Giuliani, S., Laval, H., Quartapelle, L., "Finite Element Solution of the Unsteady Navier-Stokes Equations by a Fractional Step Method," Comp. Methods Appl. Mech. Eng. 30 53-73 [1982].

Gopalakrishnan, T. C., "Comparison of Fixed and Moving Boundary Models for Coastal Circulation," Proc. First Int. Conf. Comput. Methods Flow Analysis, Vol. 1, pp. 587-593, Okayama Univ., Japan [1988].

Gresho, P. M., Chan, S. T., Upson, C., and Lee, R. L., "A Modified Finite Element Method for Solving the Time-dependent, Incompressible Navier-Stokes Equations," Int. J. Num. Meth. Fluids 4, "Part 1: Theory", pp. 557-597; "Part 2: Applications", pp. 619-640 [1984].

Gresho, P. M. and Chan, S., "Semi-Consistent Mass Matrix Techniques for Solving the Incompressible Navier-Stokes Equations," LLNL Report UCRL-99503 [1988].

Gresho, P. M. and Chan, S., "A New Semi-Implicit Method for Solving the Time-Dependent Conservation Equations for Incompressible Flow," Proc. Num. Methods in Laminar and Turbulent Flow, Pineridge Press Lt., Swansea, U. K., pp. 3-21 [1985].

Gresho, P. M. and Sani, R. L., "On Pressure Boundary Conditions for the Incompressible Navier-Stokes Equations," Int. J. Num. Methods Fluids 7 1111-1145 [1987].

Hawken, D. M., Tamaddon-Jahromi, H. R., Townsend, P. and Webster, M. F., "A Taylor-Galerkin-based Algorithm for Viscous Incompressible Flow," Int. J. Num. Methods Fluids 10 327-351 [1990].

Huffenus, J. P. and Khaletzky, D., "A Finite Element Method to Solve the Navier-Stokes Equations Using the Method of Characteristics," Int. J. Num. Methods Fluids 4 247-269 [1984].

Hughes, T. J. R., Liu, W. K., Brooks, A., "Finite Element Analysis of Incompressible Viscous Flows by the Penalty Function Formulation," J. Comp. Phys. 30 1-60 [1979].

Jackson, C. P., and Cliffe, K. A., "Mixed Interpolation in Primitive Variable Finite Element Formulation for Incompressible Flow," Int. J. Num. Methods Engn 17 1659-1688 [1981]

Kawahara, M. and Hirano, H., "A Finite Element Method for High Reynolds Number Viscous Fluid Flow Using Two Step Explicit Scheme," Int. J. Num. Methods Fluids 3 137-163 [1983].

Kovacs, A. and Kawahara, M., "A Finite Element Scheme Based on the Velocity Correction Method for the Solution of the Time-dependent Incompressible Navier-Stokes Equations," Int. J. Num. Methods Fluids 13 403-423 [1991].

Laval, H. "Taylor-Galerkin Solution of the Time-Dependent Navier-Stokes Equations," Proc. First Int. Conf. Comput. Methods Flow Analysis, pp. 414-421 Okayama Univ., Japan [1988].

Löhner, R., Morgan, K., and Zienkiewicz, O. C., "An Adaptive Finite Element Procddure for Compressible High Speed Flows," Comp. Methods Appl. Mech. Eng. 51 441-465 [1985].

Löhner, R., "An Adaptive Finite Element Scheme for Transient Problems in CFD," Comp. Methods Appl. Mech. Eng. 61 323-338 [1987].

Löhner, R., Morgan, K., Peraire, J. and Vahdati, M., "Finite Element Flux-corrected Transport (FEM-FCT) for the Euler and Navier-Stokes Equations," Int. J. Num. Methods Fluids 7 1093-1109 [1987].

Miyauchi, Y., Masuda, M., and Shimizu, M., "A Transient Finite Element Analysis of Natural Convection Around a Horizontal Hot Cylinder," Int. J. Num. Methods Fluids 3 429-443 [1983].

Oden, J. T., Strouboulis, T., and Devloo, P., "Adaptive Finite Element Methods for the Analysis of Inviscid Compressible Flow: Part 1. Fast Refinement/Unrefinement and Moving Mesh Methods for Unstructured Meshes," Comp. Meth. Appl. Mech. Eng. 59 327-362 [1986].

Pepper, D. W. and Singer, A. P, "Calculation of Convective Flow on the Person al Computer Using a Modified Finite-Element Method," Num. Heat Trans., Part A 17 379-400 [1990].

Peyret, R. and T. D. Taylor, *Computational Methods for Fluid Flow*, Springer-Verlag, [1983].

Ramaswamy, B., "Efficient Finite Element Method for Two-Dimensional Fluid Flow and Heat Transfer Problems," Num. Heat Trans., Part B 17 123-154 [1990].

Ramaswamy, B. and Kawahara, M., "Arbitrary Lagrangian-Eulerian Finite Element Method for Unsteady, Convective, Incompressible Viscous Free Surface Fluid Flow," Int. J. Num. Methods Fluids 7 1053-1075 [1987].

Sani, R. L., Gresho, P. M., Lee, R. L., Griffiths, D. F. and Engleman, M., "The Cause and Cure(!) of the Spurious Pressures Generated by Certain FEM Solutions of the Incompressible Navier-Stokes Equations: Part 2", Int. J. Num. Methods Fluids 1 171-204 [1981].

Shih, T. M., Tan, C. H. and Hwang, B. C., "Equivalence of Artificial Compressibility Method and Penalty-Function Method," Num. Heat Trans., Part B 15 127-130 [1989].

Stevens, W. N. R., "Finite Element, Stream Function-Vorticity Solution of Steady Laminar Natural Convection," Int. J. Num. Methods Fluids 2 349-366 [1982].

Thareja, R. R., Stewart, J. R., Hassan, O., Morgan, K., and Peraire, J., "A Point Implicit Unstructured Grid Solver for the Euler and Navier-Stokes Equations," Int. J. Num. Methods Fluids 9 405-425 [1989].

Yang, J. C., Chen, K. N., and Lee, H. Y., "Investigation of Use of Reach-back Characteristics Method for 2D Dispersion Equation," Int. J. Num. Methods Fluids 13 841-855 [1991].