Lecture 32

NAVIER-STOKES EQUATIONS: SPECIAL FEATURES

32.1 INTRODUCTION

Numerical simulation of the flow problems involves solution of Navier-Stokes equation. This vector equation is essentially a set of three coupled partial differential equations, and must be solved along with the continuity equation and energy equation. All these equations are coupled PDEs, and their collection is often referred to as Navier-Stokes equations in CFD literature. One remarkable feature of these equations is the similarity of their form: each of these equations can be recast in the form of the generalized transport equation which contains a time derivative term, a convective term, a diffusive terms and a source term. For compressible flows, the continuity equation, momentum equation and energy equation represent the transport equations for density, velocity and temperature respectively. Therefore, the discretization methods discussed earlier can be applied to each equation, and collection of the resulting discretized nonlinear equations can solved using a sequential iterative scheme. Thus, algorithm for numerical solution of unsteady subsonic compressible flow problem is relatively straightforward in the sense that it is an extension of the algorithms discussed in the preceding lectures for solution of generic transport equation.

Governing equations for incompressible flows are also similar in form to the generic transport equation. However, there is a small problem due to non-existence of a separate equation for pressure. In this case, there is no equation of state relating pressure, temperature and density, and continuity equation reduces to a kinematic constraint on velocity field. This requires special attention which we briefly review in this section lecture.

32.2 SPECIAL FEATURES OF NAVIER-STOKES EQUATIONS

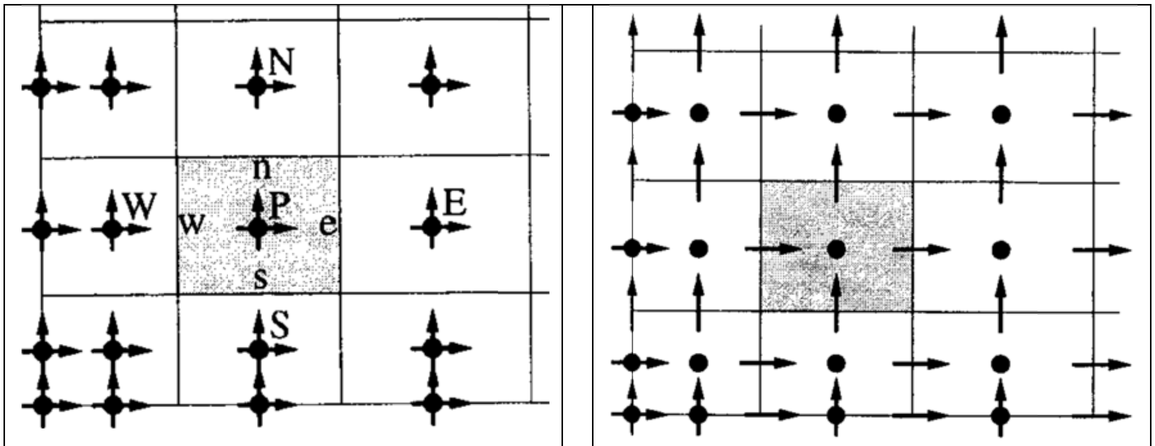
Navier-Stokes equations are governing equations for a vector field (velocity). This allows more freedom in choice of the grid used in numerical simulation. Depending on the choice of discretization scheme (finite difference, finite volume or finite element method), there are two possible choices for arrangement of the problem variables on grid nodes: (a) collocated arrangement and (b) staggered arrangement. Each of these choices has its own advantages and disadvantages which are as follows:

a) **Co-located arrangement**

If all the variables are stored at the same set of grid point, then the grid is called co-located grid (Figure 32.1a). This choice simplifies the programming and allows the use of same restriction and prolongation operators for each variable when multigrid methods are employed. Its major disadvantage is lack of pressure-velocity coupling which may lead to oscillations in pressure field (i.e. the chequer-board pattern for pressure field).

b) **Staggered arrangement**

In staggered arrangement (Figure 32.1b), pressure/scalar nodes lie at the centroids of grid cells, and velocity nodes are located at the centre of respective cell faces in case of FDM/FVM. Separate set of elements are employed in case of FEM. Disadvantage of this approach lies in elaborate book-keeping and requirement of separate set of multigrid operators for each problem variable. Biggest advantage of this approach is the strong coupling between the velocity and pressure field because of which this grid arrangement has been the most popular in CFD analysis.



(a) Co-located arrangement | (b) Staggered arrangement

Figure 32.1 Finite volume grid arrangements for pressure and velocity components for numerical solution of Navier-Stokes equations. Bullet represents computational node for scalar variables.

Another special feature relates to the mathematical nature of Navier-Stokes equations. Steady state Navier-Stokes equations are elliptic in nature, whereas the unsteady Navier-Stokes equations are parabolic in time (and elliptic in space). Due to numerical difficulties associated with solution of purely elliptic PDEs, Navier-Stokes equations are mostly solved as an unsteady problem even if the flow is steady, using a time marching scheme. Long-time solution of this transient Navier-Stokes problem yields the solution to actual steady state flow problem.

32.3 COMPUTATION OF PRESSURE

Solution of incompressible Navier-Stokes equations is complicated by the lack of an independent equation for pressure. In compressible flows, continuity equation can be used to compute density and pressure is computed from equation of state. However, for incompressible flows, continuity equation does not have a dominant variable. Thus, mass conservation is essentially a kinematic constraint on the velocity field. The way out of this difficulty is to construct the pressure field so as to guarantee satisfaction of the continuity equation (i.e. to enforce mass conservation). To achieve this objective, we combine the momentum and continuity equations to obtain an equation for pressure, which is in the form of a Poisson equation, and hence, is commonly referred to as the *pressure Poisson equation*.

To derive the pressure Poisson equation, let us rearrange the momentum as

$$\nabla p = -\left[\frac{\partial(\rho v)}{\partial t} + \nabla\cdot(\rho\mathbf{vv}) - \rho\mathbf{b} - \nabla\cdot\boldsymbol{\tau}\right] \tag{32.1}$$

In Cartesian tensor notation, the preceding equation can be re-written as

$$\frac{\partial p}{\partial x_i} = -\left[\frac{\partial(\rho v_i)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho v_i v_j - \tau_{ij}\right) \right] + \rho b_i \tag{32.2}$$

Taking divergence of the preceding equation, we get

$$\frac{\partial}{\partial x_i}\left(\frac{\partial p}{\partial x_i}\right) = -\frac{\partial}{\partial x_i}\left[\frac{\partial(\rho v_i)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho v_i v_j - \tau_{ij}\right) \right] + \frac{\partial(\rho b_i)}{\partial x_i} \tag{32.3}$$

For constant density (incompressible flow) and viscosity, viscous and unsteady terms disappear by virtue of the continuity equation. Further, body force field is usually gradient of a scalar function, and hence, its divergence also vanishes. Thus, Eq. (32.3) simplifies to

$$\frac{\partial}{\partial x_i}\left(\frac{\partial p}{\partial x_i}\right) = -\frac{\partial}{\partial x_i}\left[\frac{\partial(\rho v_i v_j)}{\partial x_j} \right] \tag{32.4}$$

Equation (32.4) represents the desired Poisson equation for pressure for incompressible flows which must be solved using an appropriate elliptic solver. Note that the Laplacian in pressure equation (9.202) is the product of the divergence operator originating from the continuity equation and the gradient operator of momentum equation. To maintain numerical consistency, it is essential that the approximation of the pressure Poisson equation must be defined as the product of the approximations of divergence and gradient operators employed in discretization of continuity and momentum equation. Otherwise satisfaction of the continuity equation cannot be guaranteed.

FURTHER READING

Ferziger, J. H. And Perić, M. (2003). *Computational Methods for Fluid Dynamics*. Springer.

Versteeg, H. K. and Malalasekera, W. M. G. (2007). *Introduction to Computational Fluid Dynamics: The Finite Volume Method*. Second Edition (Indian Reprint) Pearson Education.