Module 4: Solution of Navier-Stokes and Energy Equations for Incompressible Internal Flows

Lecture 18: Preliminaries

The Lecture Contains:

Previous Next

Solution of Navier-Stokes and Energy Equations for Incompressible Internal Flows 4.

4.1 Introduction-:

Different basic flow configurations are discribed in different parts of the heat exchangers, for example, where the fluid may flow between closely spaced plates that effectively form a duct. Although laminar duct flows do not occur as extensively as turbulent duct flows, they do occur in a number of important situations. In the case of plate-fin heat exchangers, the fin spacing is so small and the mean velocity range is such that the flows are often laminar. Conventionally, it is usual to assume that a higher heat transfer rate is achieved with turbulent flow than with laminar flow. However, sometimes it turns out that a design that involves laminar flow is the most efficient from a heat transfer viewpoint.

In this chapter, we shall consider hydrodynamically developing and fully developed flow separately in a plane channel. Flow and heat transfer in a very wide duct is considered in this section, the flow situation considered being shown in **Figure 4.1** The analysis for the pipe flow will be a straight forward extension.

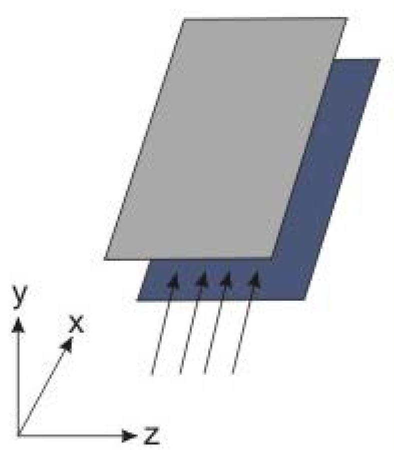

Figure 4.1: Flow in a channel

Previous Next

Navier Stokes Equations

In Cartesian coordinates, the governing equations for **incompressible three-dimensional flows** are .

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad \textbf{(4.1)}$$

$$\frac{\partial u}{\partial t}+\frac{\partial (u^2)}{\partial x}+\frac{\partial (uv)}{\partial y}+\frac{\partial (uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{1}{Re}\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right] \quad \textbf{(4.2)}$$

$$\frac{\partial v}{\partial t}+\frac{\partial (uv)}{\partial x}+\frac{\partial (v^2)}{\partial y}+\frac{\partial (vw)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{1}{Re}\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right] \quad \textbf{(4.3)}$$

$$\frac{\partial w}{\partial t}+\frac{\partial (uw)}{\partial x}+\frac{\partial (vw)}{\partial y}+\frac{\partial (w^2)}{\partial z}=-\frac{\partial p}{\partial z}+\frac{1}{Re}\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right] \quad \textbf{(4.4)}$$

In this chapter no assumption is made about the relative magnitude of the velocity components, consequently the full forms of the Navier-Stokes equations are solved. Methods described in this section will be based, basically, on finite volume and finite difference discretization and on the solution of a Poisson equation to determine the pressure. It may be mentioned that these methods use primitive variables u, v, w and ***p*** as function of x, y, z, t and Re which are preferable in flow calculations.

Staggered Grid :-

As it has been seen, the major difficulty encountered during solution of incompressible flow is the non-availability of any obvious equation for pressure. This difficulty can be resolved in the stream function-vorticity approach. **This approach loses its advantage when three-dimensional flow is computed because of the fact that a single scalar stream function does not exist in three dimensional space**

A three-dimensional problem demands a primitive-variable approach. Efforts have been made so that two as well as three-dimensional problems could be computed following a primitive variable approach without encountering non-physical wiggles in pressure distribution. As a remedy,it has been suggested to employ a different grid for each of the dependant variables. **Such a staggered grid for the dependant variables in a flow field was first used by Harlow and Welch (1965)**, in their very well known **MAC (Marker and Cell) method**. Since then,it has been used by many researchers. Specifically,**SIMPLE (Semi Implicit Method for Pressure Linked Equations)** procedure of **Patankar and Spalding (1972)** has become popular. **Figure 4.2** shows a two dimensional staggered grid where independants variables $(u_{i,j},v_{i,j}$ and $p_{i,j})$ with the identical identices staggered to one another.**Extension to three dimensions is straight-forward. Computational domain is divided into a number of cells,which are shown as "main control volume"** in **Figure 4.2.** The location of the velocity components are at the center of the cell faces to which they are normal. If a uniform grid is used, the locations are exactly at the midway between the grid points. In such cases the pressure
difference between the two adjacent cells is the driving force for the velocity component located at the interface of these cells. The finite difference approximation is now physically meaningful and the pressure field will accept a reasonable pressure distribution for a correct velocity field.

Another important advantage is that transport rates across the faces of the control volumes can be computed without interpolation of velocity components.The detailed outline of the two different solution procedures for the full **Navier-Stokes equations** with.

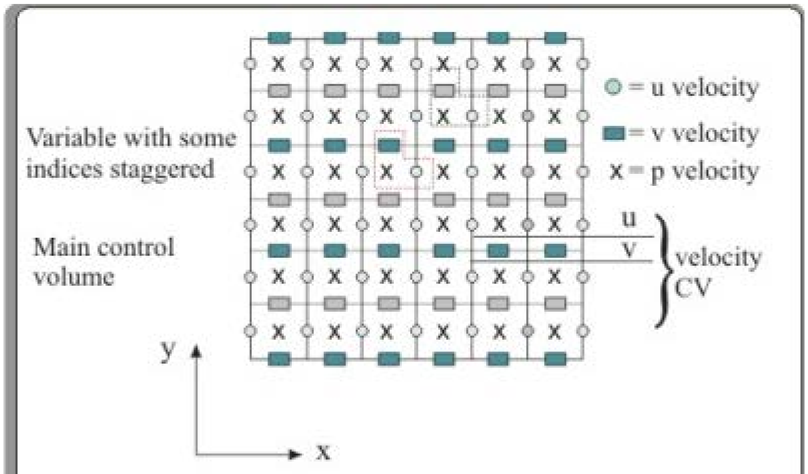

Figure 4.2: Staggered grid

Previous Next

primitive variables using staggered grid will be discussed in subsequent sections.First we shall discuss the **SIMPLE algorithm** and then the **MAC method** will be described.

Solution of the Unsteady Navier-Stokes Equations

4.4.1 Introduction to the MAC method

The **MAC method of Harlow and Welch is one of the earliest and most useful methods for solving the Navier-Stokes equations**. This method necessarily deals with a Poisson equation for the pressure and momentum equations for the computation of velocity. It was basically developed to solve problems with free surfaces, but can be applied to any incompressible fluid flow problem. A modified version of the original MAC method due to **Hirt and Cook (1972)** has been used by researchers to solve a variety of flow problems**.**The text discusses the modified MAC method and highlights the salient features of the solution algorithm so that the reader will be able to write a computer program with some confidence.

The important ideas on which the MAC algorithm is based are:

➢ **1**. Unsteady Navier-Stokes equations for incompressible flows in weak conservative form and the continuity equation are the governing equations **.**

➢ **2.** Description of the problem is elliptic in space and parabolic in time. Solution will be marched in the time direction. At each time step, a converged solution in space is obtained but this converged solution at any time step may not be the solution of the physical problem.

➢ **3.** If the problem is steady, in its physical sense, then after a finite number of steps in time direction, two consecutive time-steps will show identical solutions. However, in a machine-computation this is not possible hence a very small upper bound, say,**"STAT"** is predened. Typically,STAT may be chosen between 10^{-3} and 10^{-5}. If the maximum discrepancy of any of the velocity components for two consecutive time steps for any location over the entire space does not exceed STAT, then it can be said that the steady solution has been evolved.

4. If the physical problem is basically unsteady in nature, the aforesaid maximum discrepancy of any dependant variable for two consecutive time steps will never be less than **STAT**. However, for such a situation, a specified velocity component can be stored over a long duration of time and plot of the velocity component against time **(often called as signal)** depicts the character of the flow. Such a flow may be labeled simply as "**unsteady**".

5. With the help of the momentum equations, we compute explicitly a provisional value of the velocity components for the next time step.

Consider the weak conservative form of the nondimensional momentum equation x **in the x** direction:

$$\frac{\partial u}{\partial t}+\frac{\partial (u^2)}{\partial x}+\frac{\partial (uv)}{\partial y}+\frac{\partial (uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{1}{Re}\nabla^2 u \qquad \textbf{(4.5)}$$

It is assumed that at $t = n^{th}$ level, we have a converged solution. Then for the next time step

$$\frac{\tilde{u}^{n+1}_{i,j,k}-u^{n}_{i,j,k}}{\Delta t}=[\text{CONDIFU-DPDX}]^{n}_{i,j,k} \qquad \textbf{(4.6)}$$

$$\tilde{u}^{n+1}_{i,j,k}=u^{n}_{i,j,k}+\Delta t\ \ [\text{CONDIFU-DPDX}]^{n}_{i,j,k} \qquad \textbf{(4.7)}$$

Previous Next

$[CONDIFU - DPDX]^n_{i,j,k}$ consists of convective and diusive terms,and the pressure gradient. Similarly,the provisional values for $\tilde{v}^{n+1}_{i,j,k}$ and $\tilde{w}^{n+1}_{i,j,k}$ a can be explicitly computed.These explicitly advanced velocity components may not constitute a realistic flow eld. A divergence free velocity field has to exist in order to describe a plausible incompressible flow situation. Now, with these provisional $\tilde{u}^{n+1}_{i,j,k}$, $\tilde{v}^{n+1}_{i,j,k}$ and $\tilde{w}^{n+1}_{i,j,k}$ values,continuity equation is evaluated in each cell.If $(\nabla \cdot V)$ produces a nonzero value,there must be some amount of mass accumulation or annihilation in each cell which is not physically possible.Therefore the pressure at any cell is directly linked with the value of the $(\nabla \cdot V)$ of that cell. Now, on one hand the pressure has to be calculated with the help of the nonzero divergence value and on the other, the velocity components have to be adjusted.The correction procedure continues through an iterative cycle till the divergence free velocity field is ensured. Details of the procedure will be discussed in the subsequent section.

6.Boundary conditions are to be applied after each explicit evaluation for the time step is accomplished. Since the governing equations are elliptic in space, boundary conditions on all confining surfaces are required. Moreover, the boundary conditions are also to be applied after every pressure-velocity iteration. **The five principal kinds of boundary conditions to be considered are rigid no-slip walls, free-slip walls, in flow and out flow boundaries, and periodic (repeating) boundaries.**

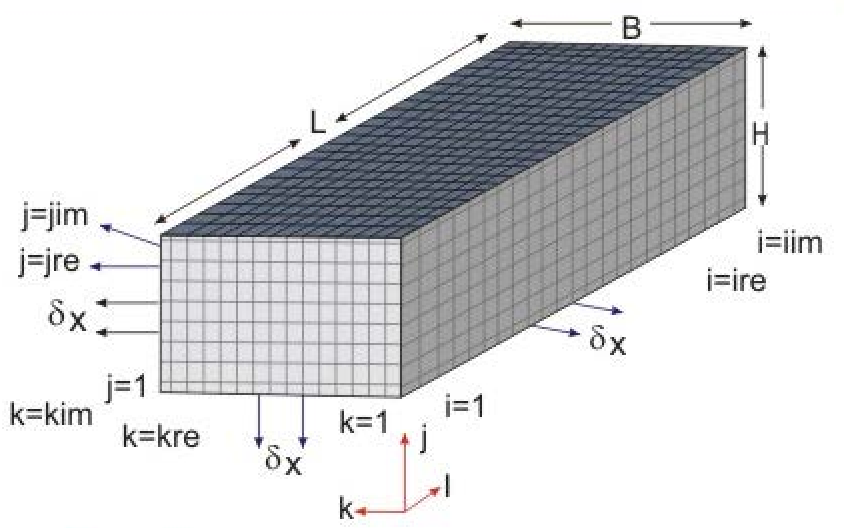

Figure 4.3: Discretization of a Three-dimensional domain