Chapter 1

INTRODUCTION

1.1 Motivation

The material in this book originated from attempts to understand and systemize numerical solution techniques for the partial differential equations governing the physics of fluid flow. As time went on and these attempts began to crystallize, underlying constraints on the nature of the material began to form. The principal such constraint was the demand for unification. Was there one mathematical structure which could be used to describe the behavior and results of most numerical methods in common use in the field of fluid dynamics? Perhaps the answer is arguable, but the authors believe the answer is affirmative and present this book as justification for that belief. The mathematical structure is the theory of linear algebra and the attendant eigenanalysis of linear systems.

The ultimate goal of the field of computational fluid dynamics (CFD) is to understand the physical events that occur in the flow of fluids around and within designated objects. These events are related to the action and interaction of phenomena such as dissipation, diffusion, convection, shock waves, slip surfaces, boundary layers, and turbulence. In the field of aerodynamics, all of these phenomena are governed by the compressible Navier-Stokes equations. Many of the most important aspects of these relations are nonlinear and, as a consequence, often have no analytic solution. This, of course, motivates the numerical solution of the associated partial differential equations. At the same time it would seem to invalidate the use of linear algebra for the classification of the numerical methods. Experience has shown that such is not the case.

As we shall see in a later chapter, the use of numerical methods to solve partial differential equations introduces an approximation that, in effect, can change the form of the basic partial differential equations themselves. The new equations, which are the ones actually being solved by the numerical process, are often referred to as the modified partial differential equations. Since they are not precisely the same as the original equations, they can, and probably will, simulate the physical phenomena listed above in ways that are not exactly the same as an exact solution to the basic partial differential equation. Mathematically, these differences are usually referred to as truncation errors. However, the theory associated with the numerical analysis of fluid mechanics was developed predominantly by scientists deeply interested in the physics of fluid flow and, as a consequence, these errors are often identified with a particular physical phenomenon on which they have a strong effect. Thus methods are said to have a lot of "artificial viscosity" or said to be highly dispersive. This means that the errors caused by the numerical approximation result in a modified partial differential equation having additional terms that can be identified with the physics of dissipation in the first case and dispersion in the second. There is nothing wrong, of course, with identifying an error with a physical process, nor with deliberately directing an error to a specific physical process, as long as the error remains in some engineering sense "small". It is safe to say, for example, that most numerical methods in practical use for solving the nondissipative Euler equations create a modified partial differential equation that produces some form of dissipation. However, if used and interpreted properly, these methods give very useful information.

Regardless of what the numerical errors are called, if their effects are not thoroughly understood and controlled, they can lead to serious difficulties, producing answers that represent little, if any, physical reality. This motivates studying the concepts of stability, convergence, and consistency. On the other hand, even if the errors are kept small enough that they can be neglected (for engineering purposes), the resulting simulation can still be of little practical use if inefficient or inappropriate algorithms are used. This motivates studying the concepts of stiffness, factorization, and algorithm development in general. All of these concepts we hope to clarify in this book.

1.2 Background

The field of computational fluid dynamics has a broad range of applicability. Independent of the specific application under study, the following sequence of steps generally must be followed in order to obtain a satisfactory solution.

1.2.1 Problem Specification and Geometry Preparation

The first step involves the specification of the problem, including the geometry, flow conditions, and the requirements of the simulation. The geometry may result from measurements of an existing configuration or may be associated with a design study. Alternatively, in a design context, no geometry need be supplied. Instead, a set of objectives and constraints must be specified. Flow conditions might include, for example, the Reynolds number and Mach number for the flow over an airfoil. The requirements of the simulation include issues such as the level of accuracy needed, the turnaround time required, and the solution parameters of interest. The first two of these requirements are often in conflict and compromise is necessary. As an example of solution parameters of interest in computing the flowfield about an airfoil, one may be interested in i) the lift and pitching moment only, ii) the drag as well as the lift and pitching moment, or iii) the details of the flow at some specific location.

1.2.2 Selection of Governing Equations and Boundary Conditions

Once the problem has been specified, an appropriate set of governing equations and boundary conditions must be selected. It is generally accepted that the phenomena of importance to the field of continuum fluid dynamics are governed by the conservation of mass, momentum, and energy. The partial differential equations resulting from these conservation laws are referred to as the Navier-Stokes equations. However, in the interest of efficiency, it is always prudent to consider solving simplified forms of the Navier-Stokes equations when the simplifications retain the physics which are essential to the goals of the simulation. Possible simplified governing equations include the potential-flow equations, the Euler equations, and the thin-layer Navier-Stokes equations. These may be steady or unsteady and compressible or incompressible. Boundary types which may be encountered include solid walls, inflow and outflow boundaries, periodic boundaries, symmetry boundaries, etc. The boundary conditions which must be specified depend upon the governing equations. For example, at a solid wall, the Euler equations require flow tangency to be enforced, while the Navier-Stokes equations require the no-slip condition. If necessary, physical models must be chosen for processes which cannot be simulated within the specified constraints. Turbulence is an example of a physical process which is rarely simulated in a practical context (at the time of writing) and thus is often modelled. The success of a simulation depends greatly on the engineering insight involved in selecting the governing equations and physical models based on the problem specification.

1.2.3 Selection of Gridding Strategy and Numerical Method

Next a numerical method and a strategy for dividing the flow domain into cells, or elements, must be selected. We concern ourselves here only with numerical methods requiring such a tessellation of the domain, which is known as a grid, or mesh.

Many different gridding strategies exist, including structured, unstructured, hybrid, composite, and overlapping grids. Furthermore, the grid can be altered based on the solution in an approach known as solution-adaptive gridding. The numerical methods generally used in CFD can be classified as finite-difference, finite-volume, finite-element, or spectral methods. The choices of a numerical method and a gridding strategy are strongly interdependent. For example, the use of finite-difference methods is typically restricted to structured grids. Here again, the success of a simulation can depend on appropriate choices for the problem or class of problems of interest.

1.2.4 Assessment and Interpretation of Results

Finally, the results of the simulation must be assessed and interpreted. This step can require post-processing of the data, for example calculation of forces and moments, and can be aided by sophisticated flow visualization tools and error estimation techniques. It is critical that the magnitude of both numerical and physical-model errors be well understood.

1.3 Overview

It should be clear that successful simulation of fluid flows can involve a wide range of issues from grid generation to turbulence modelling to the applicability of various simplified forms of the Navier-Stokes equations. Many of these issues are not addressed in this book. Some of them are presented in the books by Anderson, Tannehill, and Pletcher [1] and Hirsch [2]. Instead we focus on numerical methods, with emphasis on finite-difference and finite-volume methods for the Euler and Navier-Stokes equations. Rather than presenting the details of the most advanced methods, which are still evolving, we present a foundation for developing, analyzing, and understanding such methods.

Fortunately, to develop, analyze, and understand most numerical methods used to find solutions for the complete compressible Navier-Stokes equations, we can make use of much simpler expressions, the so-called “model” equations. These model equations isolate certain aspects of the physics contained in the complete set of equations. Hence their numerical solution can illustrate the properties of a given numerical method when applied to a more complicated system of equations which governs similar physical phenomena. Although the model equations are extremely simple and easy to solve, they have been carefully selected to be representative, when used intelligently, of difficulties and complexities that arise in realistic two- and three-dimensional fluid flow simulations. We believe that a thorough understanding of what happens when numerical approximations are applied to the model equations is a major first step in making confident and competent use of numerical approximations to the Euler and Navier-Stokes equations. As a word of caution, however, it should be noted that, although we can learn a great deal by studying numerical methods as applied to the model equations and can use that information in the design and application of numerical methods to practical problems, there are many aspects of practical problems which can only be understood in the context of the complete physical systems.

1.4 Notation

The notation is generally explained as it is introduced. Bold type is reserved for real physical vectors, such as velocity. The vector symbol $\vec{\ }$ is used for the vectors (or column matrices) which contain the values of the dependent variable at the nodes of a grid. Otherwise, the use of a vector consisting of a collection of scalars should be apparent from the context and is not identified by any special notation. For example, the variable u can denote a scalar Cartesian velocity component in the Euler and Navier-Stokes equations, a scalar quantity in the linear convection and diffusion equations, and a vector consisting of a collection of scalars in our presentation of hyperbolic systems. Some of the abbreviations used throughout the text are listed and defined below.

PDE	Partial differential equation
ODE	Ordinary differential equation
OΔE	Ordinary difference equation
RHS	Right-hand side
P.S.	Particular solution of an ODE or system of ODE's
S.S.	Fixed (time-invariant) steady-state solution
k-D	k-dimensional space
$\left(\vec{bc}\right)$	Boundary conditions, usually a vector
O(α)	A term of order (i.e., proportional to) α