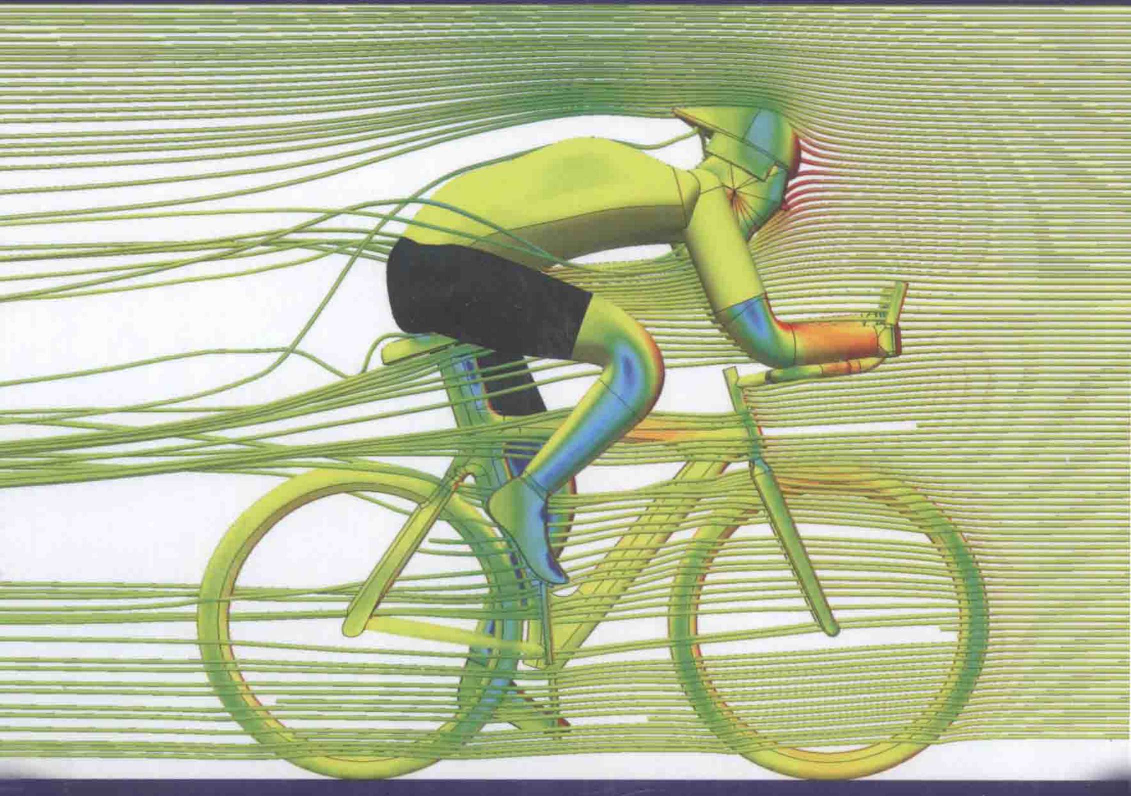
ESSENTIAL COMPUTATIONAL FLUID DYNAMICS



OLEG ZIKANOV

ESSENTIAL COMPUTATIONAL FLUID DYNAMICS

Oleg Zikanov

WILEY

JOHN WILEY & SONS, INC.

This book is printed on acid-free paper. ♾

Published by John Wiley & Sons, Inc., Hoboken, New Jersey

Published simultaneously in Canada

Library of Congress Cataloging-in-Publication Data:
Zikanov, Oleg.
Essential computational fluid dynamics / Oleg Zikanov.
p. cm.
Includes bibliographical references and index.
ISBN 978-0-470-42329-5 (cloth)
1. Fluid dynamics–Mathematics. I. Title.
QA911.Z55 2010
532′.0501515–dc22

2009042596

ISBN: 978-0-470-42329-5

Printed in the United States of America

10 9 8 7 6 5 4 3 2 1

To Elena

PREFACE

This book is a complete and self-contained introduction into computational fluid dynamics and heat transfer, commonly abbreviated as CFD. The text addresses this subject on the very basic level suitable for a first course of CFD taught to beginning graduate or senior undergraduate students. No prior knowledge of CFD is assumed on the part of the reader.

To appreciate the purpose and flavor of the book, we have to consider the major shift that currently occurs in the scope and character of CFD applications. From being a primarily research discipline just 20 years ago, CFD has transformed into a tool of everyday engineering practice. It would be safe to say that, worldwide, tens of thousands of engineers are directly employed to run CFD computations at companies or consulting firms. Many others encounter CFD at some stages of their work.

Unlike solution of research problems, CFD analysis in industrial environment does not, typically, involve development of new algorithms. Instead, one of the general purpose codes is used. Such codes, nowadays, tend to provide a fusion of all the necessary tools: equation solver, mesh generator, turbulence and multiphysics models, and modules for post-processing and parallel computations. Two key factors contribute to the success in applying such codes: (1) Understanding of physical and engineering aspects of the analyzed process; and (2) Ability to conduct the CFD analysis properly, in a way that guarantees an accurate and efficient solution.

I recognized the need for a new textbook when I was teaching the graduate and senior undergraduate courses in CFD at the Department of Mechanical Engineering of the University of Michigan–Dearborn. The majority of our graduate students are either working engineers or researchers in applied engineering fields. The undergraduate students tend to pursue industrial employment after graduation. Potential future exposure of our students to CFD is often limited to the use of general purpose codes. To respond to their needs, the instruction is focused on two areas: the fundamentals of the method (what we call the *essential CFD*) and the correct way of conducting the analysis using readily available software. A survey of the existing texts on CFD, although revealing many excellent research-oriented texts, does not reveal a book that fully corresponds to this concept.

A comment is in order regarding the bias of the text. All CFD texts are, to some degree, biased in correspondence to the chosen audience and research interests of the authors. More weight is given to some of the methods (finite difference, finite element, spectral, etc.) and to some of the fields of application (heat transfer, incompressible fluid dynamics, or gas dynamics). The choices made in this book reflect the assumption of mechanical, chemical, and civil engineering students as the target audience rather than aerospace engineering students, and the intended use of the text for applied CFD instruction. The focus is on the finite difference and finite volume methods. The finite element and spectral techniques are introduced only briefly. Also, somewhat more attention is given to numerical methods for incompressible fluid dynamics and heat transfer than for compressible flows.

The text can be used in combination with exercises in practical CFD analysis. As an example, our course at the University of Michigan–Dearborn is divided into two parts. The first part (about 60 percent of the total course time) is reserved for classroom instruction of the basic methods of CFD. It covers Part I, "Fundamentals," and Part II, "Methods." It includes a simple programming project (solving a one-dimensional heat or wave equation). The remainder of the course includes exercises with a CFD software and parallel discussion of the topics of Part III, "Art of CFD" dealing with turbulence modeling, computational grids, and rules of good CFD practice. This part is conducted in a computer laboratory and includes a project in which students perform a full-scale CFD analysis.

Acknowledgments: It is a pleasure to record my gratitude to many people who made writing this book possible. This includes generations of students at the University of Michigan–Dearborn, who suffered through the first iterations of the text and provided priceless feedback. I wish to thank friends and colleagues who read the manuscript and gave their insightful and constructive suggestions: Thomas Boeck, Dmitry Krasnov, Svetlana Poroseva, Tariq Shamim, Olga Shishkina, Sergey Smolentsev, Axelle Viré, and Anatoly Vorobev. The first serious attempt to write the book was undertaken during a sabbatical stay at the Ilmenau University of Technology. I appreciate the hospitality of Andre Thess and support by the German Science Foundation (DFG) that made this possible. Finally, and above all, I would like to thank my wife, Elena, and my children, Kirill and Sophia, for their understanding and support during the many hours it took to complete this book.

CONTENTS

1

WHAT IS CFD?

1.1 INTRODUCTION

We start with a definition:

> CFD (computational fluid dynamics) is a set of numerical methods applied to obtain approximate solutions of problems of fluid dynamics and heat transfer.

According to this definition, CFD is not a science by itself but a way to apply the methods of one discipline (numerical analysis) to another (heat and mass transfer). We will deal with details later. Right now, a brief discussion is in order of why exactly we need CFD.

A distinctive feature of the science of fluid flow and heat and mass transfer is the approach it takes toward description of physical processes. Instead of bulk properties, such as momentum or angular momentum of a body in mechanics or total energy or entropy of a system in thermodynamics, the analysis focuses on *distributed properties*. We try to determine entire *fields* such as temperature $T(\boldsymbol{x}, t)$ velocity $\boldsymbol{v}(\boldsymbol{x}, t)$, density $\rho(\boldsymbol{x}, t)$, etc.[1] Even when an integral characteristic, such as the friction coefficient or the net rate of heat transfer, is the ultimate goal of analysis, it is derived from distributed fields.

The approach is very attractive by virtue of the level of details it provides. Evolution of the entire temperature distribution within a body can

[1]Throughout the book, we will use $\boldsymbol{x} = (x, y, z)$ for the vector of space coordinate and t for time.

be determined. Internal processes of a fluid flow such as motion, rotation, and deformation of minuscule fluid particles can be taken into account. Of course, the opportunities come at a price, most notably in the form of dramatically increased complexity of the governing equations. Except for a few strongly simplified models, the equations for distributed properties are *partial differential equations*, often nonlinear.

As an example of complexity, let us consider a seemingly simple task of mixing and dissolving sugar in a cup of hot coffee. An innocent question of how long or how many rotations of a spoon would it take to completely dissolve the sugar leads to a very complex physical problem that includes a possibly turbulent two-phase (coffee and sugar particles) flow with a chemical reaction (dissolving). Heat transfer (within the cup and between the cup and surroundings) may also be of importance because temperature affects the rate of the reaction. No simple solution of the problem exists. Of course, we can rely on the experience acquired after repeating the process daily (perhaps more than once) for many years. We can also add a couple of extra, possibly unnecessary, stirs. If, however, the task in question is more serious—for example, optimizing an oil refinery or designing a new aircraft—relying on everyday experience or excessive effort is not an option. We must find a way to *understand* and *predict* the process.

Generally, we can distinguish three approaches to solving fluid flow and heat transfer problems:

1. *Theoretical approach*—using governing equations to find analytical solutions
2. *Experimental approach*—staging a carefully designed experiment using a model of the real object
3. *Numerical approach*—using computational procedures to find a solution

Let's look at these approaches in more detail.

Theoretical approach. The approach has a crucial advantage of providing exact solutions. Among the disadvantages, the most important is that analytical solutions are only possible for a very limited class of problems, typically formulated in an artificial, idealized way. One example is the Poiseuille solution for a flow in an infinitely long pipe (see Figure 1.1). The steady-state laminar velocity profile is

$$U(r) = \frac{r^2 - R^2}{4\mu}\frac{dp}{dx},$$

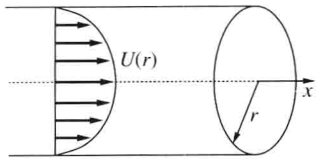


Figure 1.1 Laminar flow in an infinite pipe.

where U is the velocity, R is the pipe radius, dp/dx is the constant pressure gradient that drives the flow, and μ is the dynamic viscosity of the fluid. On the one hand, the solution is, indeed, simple and gives insight into the nature of flows in pipes and ducts, so its inclusion into all textbooks of fluid dynamics is not surprising. On the other hand, the solution is correct only if the pipe is infinitely long,[2] temperature is constant, and the fluid is perfectly incompressible. Furthermore, even if we were able to build such a pipe and find a useful application for it, the solution would be correct only at Reynolds numbers $Re = UR\rho/\mu$ (ρ is the density of the fluid) that are below approximately 2,000. Above this limit, the flow would assume fully three-dimensional and time-dependent turbulent form, for which no analytical solution is possible.

It can also be noted that derivation of analytical solutions often requires substantial mathematical skills, which are not among the strongest traits of many modern engineers and scientists, especially if compared to the situation of 30 or 40 years ago. Several reasons can be named for the deterioration of such skills, one, no doubt, being development of computers and numerical methods, including the CFD.

Experimental approach. Well-known examples are the wind tunnel experiments, which help to design and optimize the external shapes of airplanes (also of ships, buildings, and other objects). Another example is illustrated in Figure 1.2. The main disadvantages of the experimental approach are the technical difficulty (sometimes it takes several years before an experiment is set up and all technical problems are resolved) and high cost.

Numerical (computational) approach. Here, again, we employ our ability to describe almost any fluid flow and heat transfer process as a solution of a set of partial differential equations. An approximation to this solution is found in the result of a computational procedure. This approach is not problem-free, either. We will discuss the problems throughout the book.

[2]In practice, the solution is considered to be a good approximation for laminar flows in pipes at sufficiently large distance (dependent on the Reynolds number but, at least few tens of diameters) from the entrance.

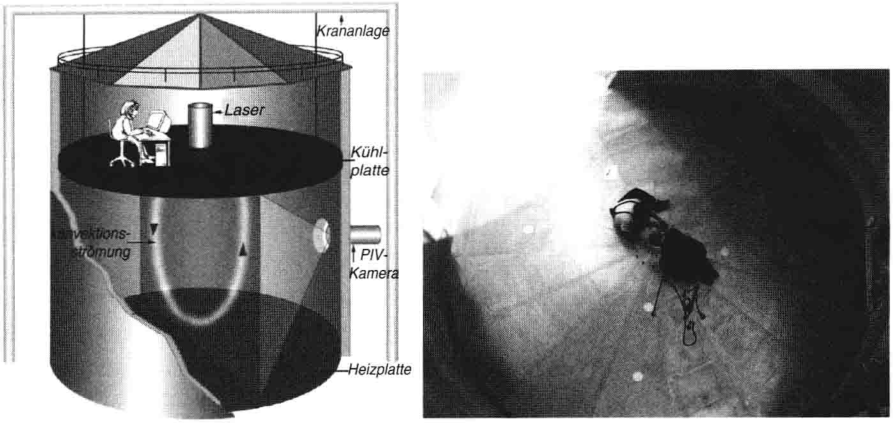


Figure 1.2 The experiment for studying thermal convection at the Ilmenau University of Technology, Germany (courtesy of A. Thess). Turbulent convection similar to the convection observed in the atmosphere of Earth or Sun is simulated by air motion within a large barrel with thermally insulated walls and uniformly heated bottom.

The computational approach, however, beats the analytical and experimental methods in some very important aspects: universality, flexibility, accuracy, and cost.

1.2 BRIEF HISTORY OF CFD

The history of CFD is a fascinating subject, which, unfortunately, we can only touch in passing. The idea to calculate approximate solutions of differential equations describing fluid flows and heat transfer is relatively old. It is definitely older than computers themselves. Development of numerical methods for solving ordinary and partial differential equations started in the first half of the twentieth century. The computations at that time required use of tables and dull mechanical work of dozens, if not hundreds, of people. No wonder that only the most important (primarily military-related) problems were addressed and only simple, one-dimensional equations were solved.

Invention and subsequent fast development of computers (see Figure 1.3) opened a wonderful possibility of performing millions—and then millions of millions—of arithmetic operations in a matter of seconds. This caused a rapid growth of the efforts to develop and apply methods of numerical simulations. Again, military applications, such as modeling shock waves from an explosion or a flow past a hypersonic