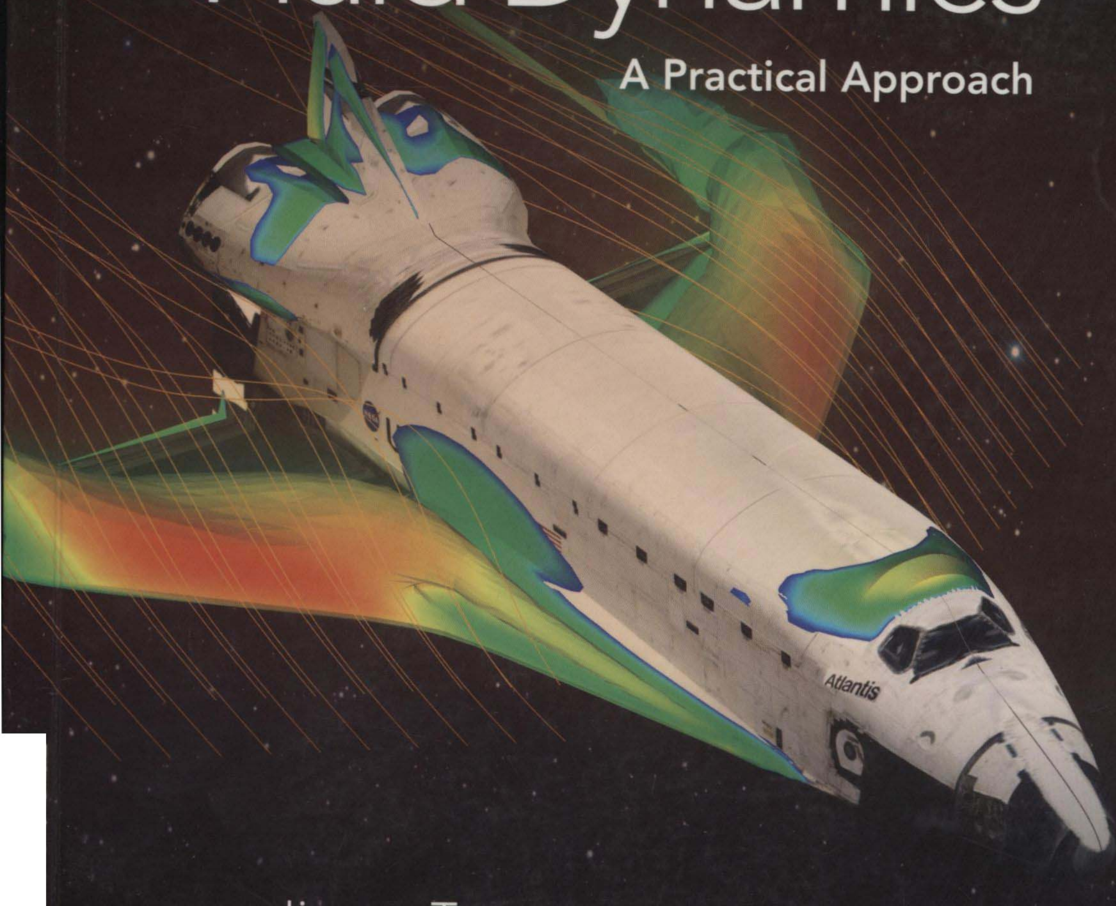
Computational Fluid Dynamics

SECOND EDITION

A Practical Approach



Jiyuan Tu
Guan-Heng Yeoh
Chaoqun Liu

Computational Fluid Dynamics

A Practical Approach

Second Edition

Jiyuan Tu
RMIT University, Australia

Guan-Heng Yeoh
Australian Nuclear Science
and Technology Organisation
University of New South Wales, Australia

Chaoqun Liu
University of Texas at Arlington

ELSEVIER

AMSTERDAM • BOSTON • HEIDELBERG • LONDON
NEW YORK • OXFORD • PARIS • SAN DIEGO
SAN FRANCISCO • SINGAPORE • SYDNEY • TOKYO

Butterworth-Heinemann is an imprint of Elsevier

Butterworth-Heinemann is an imprint of Elsevier
225 Wyman Street, Waltham, MA 02451, USA
The Boulevard, Langford Lane, Kidlington, Oxford, OX5 1 GB, UK

Library of Congress Cataloging-in-Publication Data
Application submitted.

British Library Cataloguing-in-Publication Data
A catalogue record for this book is available from the British Library.

ISBN: 978-0-08-098243-4

For information on all Butterworth-Heinemann publications,
visit our website: http://store.elsevier.com

Printed in the United Kingdom
12 13 14 15 16 10 9 8 7 6 5 4 3 2 1

Preface to the First Edition

Computational fluid dynamics (CFD), once the domain of academics, postdoctoral researchers, or trained specialists, is now progressively becoming more accessible to graduate engineers for research and development as well as design-oriented tasks in industry. Mastery of CFD in handling complex flow and heat industrial problems is becoming ever more important. Competency in such a skill certainly brings about a steep learning curve for practicing engineers, who constantly face extreme challenges to come up with solutions to fluid flow and heat transfer problems without *a priori* knowledge of the basic concepts and fundamental understanding of fluid mechanics and heat transfer.

Today's engineers are almost certainly geared more toward the use of commercial CFD codes, such as ANSYS-CFX, ANSYS-FLUENT, or STAR-CD. Without proper guidance, the use of these software packages poses risks likened to placing potent weaponry in the hands of poorly trained soldiers. There is every possibility of users with inadequate training causing more harm than good through flawed interpretations of results produced through such packages. This makes it ever more important that a sound knowledge of CFD be acquired. Furthermore, a changing workplace environment has imposed constraints on users in discerning the pitfalls of CFD by osmosis and through frequent failures. The number of users who have had the luxury of being fully equipped and who are conscious of the limitations of CFD from their own experiences is fast dwindling.

The purpose of this book is to offer CFD users a *suitable text* pitched at *the right level* of assumed knowledge. CFD is a mathematically sophisticated discipline and the authors' aim in this book has been to provide simple-to-understand descriptions of fundamental CFD theories, basic CFD techniques, and practical guidelines. It has never been our aim to overwhelm the reader with excessive mathematical and theoretical illustrations of computational techniques. Every effort has been made to discuss the material in a style to capture the reader's attention. The dominant feature of the present book is to maintain practicality in understanding CFD. In our lecturing experience on CFD, we have identified what it takes to present elementary concepts to initiate the student. This book incorporates specially designed *intuitive* and *systematic* worked-out CFD examples to enhance the learning process, and it provides students with examples for practice to better comprehend the basic principles. It is hoped that this approach will accomplish the purpose of offering techniques to *beginners* who are more focused on the *engineering practice* of CFD.

The basic structure of this book is as follows:

Chapter 1 presents an introduction to computational fluid dynamics and is specifically designed to provide the reader with an overview of CFD and its entailed advantages, the range of applications as a research tool on various facets of industrial problems, and the future use of CFD.

Chapter 2 aims to cultivate a sense of curiosity for the first-time user on how a CFD problem is currently handled and solved. The reader will benefit through guidance of these basic processes using any commercial, shareware and in-house CFD codes. More importantly, Chapter 2 serves as a guidepost for the reader to other chapters relating to fundamental knowledge of CFD. Chapter 2 has a unique design compared with many traditional CFD presentations.

The basic thoughts and philosophy associated with CFD, along with an extensive discussion of the governing equations of fluid dynamics and heat transfer, are treated in Chapter 3. It is vitally important that the reader can fully appreciate, understand, and feel comfortable with the basic physical equations and underlying principles of this discipline, as they are its lifeblood. By working through the worked-out examples, the reader will have a better understanding of the equations governing the conservation of mass, momentum, and energy.

Computational solutions are obtained in two stages. The first stage deals with numerical discretization, which is examined in Chapter 4. Here, the basic numerics are illustrated with popular discretization techniques, such as the finite-difference and finite-volume methods (adopted in the majority of commercial codes) for solving flow problems. The second stage deals with the specific techniques for solving algebraic equations. The pressure–velocity coupling scheme (SIMPLE and its derivatives) in this chapter forms the information core of the book. This scheme invariably constitutes the basis of most commercial CFD codes through which simulations of complex industrial problems have been successfully made.

The numerical concepts of stability, convergence, consistency, and accuracy are discussed in Chapter 5. As an understanding of the fundamental equations of fluid flow and heat transfer is the essence of CFD, it follows that the understanding of the techniques of achieving a CFD solution is the resultant substantive. This chapter will enable the reader to better assess the results produced when different numerical methodologies are applied.

The authors have included turbulence modeling in CFD, a subject not ordinarily treated in a book of this nature, but after careful consideration, we have felt it imperative to include it since *real-world applications of CFD* are turbulent in nature after all. In Chapter 6, the authors have therefore devised some practical guidelines for the reader to better comprehend turbulence modeling and other models commonly applied. The authors have also carefully designed worked-out examples that will assist students in the understanding of the complex modeling concept.

An increasing number of books and journals covering different aspects of CFD in mathematically abstruse terms are readily available, mainly for specialists associated with industry. It follows that it is more helpful to include in Chapter 7 illustrations of the power of CFD through a set of industrially relevant applications on a significant range of engineering disciplines. Special efforts have been made in this chapter to stimulate the inquisitive minds of the reader through exposition of some pioneering applications.

Although detailed treatment of advanced CFD techniques is usually outside the scope of a book of this nature, we have offered a general introduction to the basic concepts in Chapter 8, hoping, in the process, to reap the benefits of whetting the readers' appetite for more to come in the evolutionary use of CFD in any new emerging areas of science and engineering.

Jiyuan Tu
Guan-Heng Yeoh
Chaoqun Liu

Preface to the Second Edition

The acceptance of the first edition of our book by the CFD community has certainly been overwhelming and most welcomed. We were extremely pleased by the positive feedback received even within the short period since its publication. In responding to the numerous comments, the second edition aims to further enhance and update the fast-growing subject of CFD, including significant developments and important applications. In order not to stray away from our primary focus of offering CFD users a *suitable text* that is pitched at *the right level* of assumed knowledge, the structure and systematic approach of the first edition have been retained.

In the treatment of the fundamental physics of fluid flows, we have added the generic form of the equations pertaining to compressible flows, which are in retrospect an extension to the incompressible form of the equations for CFD.

At the time of the writing of the first edition, we focused predominantly on the description of the most popular discretization approaches in CFD, the finite-difference and finite-volume methods. Recognizing that other discretization methods, such as the finite-element and spectral methods, are still available in the mainstream of CFD, we have provided a summary of the basic ideas underpinning the use of these methods to solve the fluid-flow equations. To reflect the iterative approach that is also commonly being adopted to solve systems of discretized equations in commercial CFD codes, we have written a section in Chapter 4 dedicated to the multi-grid method.

In the first edition, we also identified a number of key sectors where CFD has been firmly established. They are aerospace, biomedical science and engineering, chemical and mineral processing, civil and environmental engineering, power generation, and sports. In this second edition, we add the application of CFD in the areas of metallurgy and nuclear safety. Considering the wide spread of CFD throughout a number of significant engineering areas, the proper handling of complex geometries becomes ever more important in view of different types of meshing approaches that can be employed. Discussions on key aspects of structured, body-fitted, and unstructured meshes have been provided within the practical guidelines of grid generation.

Finally, an alternative numerical approach based on the discrete element method has been added to the growing list of advanced topics in CFD. For instructors adopting this text for use in their courses, solutions to end-of-chapter problems and a set of PowerPoint slides are available by registering at www.textbooks.elsevier.com.

Jiyuan Tu
Guan-Heng Yeoh
Chaoqun Liu

Acknowledgments

The material presented in this book has been partly accumulated from teaching the course *Introduction to Computational Fluid Dynamics* for senior undergraduate students at the School of Aerospace, Mechanical and Manufacturing Engineering at the Royal Melbourne Institute Technology University, Australia, as well as the course *Computational Engineering* at the School of Mechanical and Manufacturing Engineering, University of New South Wales, Australia. We thank the students who have taken the course for providing us with feedback in designing particular project topics and in aiding understanding of the subject. The authors also thank many research students and colleagues who have generously assisted us in various ways.

For the first edition, we thank Jonathan Simpson, senior commissioning editor, and his colleagues at Elsevier Science & Technology who have offered us immense help in both their academic elucidation and their professional skills in the publication process. Our special thanks are also given to Dr. Risa Robinson, Associate Professor of Mechanical Engineering at the Rochester Institute of Technology, for reviewing the whole text within a very short period of time and giving us invaluable suggestions and comments that we have incorporated in this book.

We thank Joe Hayton and Fiona Geraghty from Elsevier Science and Technology for initiating and managing the project for the second edition of our book.

Dr. Tu expresses his deep gratitude to his wife, Xue, and his son, Tian, who have provided their unflinching support in the preparation and writing of the text.

Dr. Yeoh acknowledges the untiring support of his wife, Natalie, and his daughters, Genevieve, Ellana, and Clarissa, and thanks them for their understanding and encouragement during the seemingly unending hours spent in preparing and writing.

Dr. Liu acknowledges the strong support and encouragement received from his wife, Weilan, his daughter, Haiyan, and his son, Haifeng, during the preparation of the text.

To all who have been involved, we extend our deepest, heartfelt appreciation.

Contents

Chapter 1

Introduction

1.1 WHAT IS COMPUTATIONAL FLUID DYNAMICS?

Computational fluid dynamics has certainly come of age in industrial applications and academic research. In the beginning, this popular field of study, usually referred to by its acronym CFD, was only known in the high-technology engineering areas of aeronautics and astronautics, but now it is becoming a rapidly adopted methodology for solving complex problems in modern engineering practice. CFD, which is derived from the disciplines of fluid mechanics and heat transfer, is also finding its way into important uncharted areas, especially in process, chemical, civil, and environmental engineering. Construction of new and improved system designs and optimization carried out on existing equipment through computational simulations are resulting in enhanced efficiency and lower operating costs. With the concerns about global warming and the world's increasing population, engineers in power-generation industries are heavily relying on CFD to reduce development and retrofitting costs. These computational studies are currently being performed to address pertinent issues relating to technologies for clean and renewable power as well as for meeting strict regulatory challenges, such as emissions control and substantial reduction of environmental pollutants.

Nevertheless, the basic question remains: What is *computational fluid dynamics*? In retrospect, it has certainly evolved, integrating not only the disciplines of fluid mechanics with mathematics but also computer science, as illustrated in Figure 1.1. Let's briefly discuss each of these individual disciplines. Fluid mechanics is essentially the study of fluids, either in motion (*fluid in dynamic mode*) or at rest (*fluid in stationary mode*). CFD is particularly dedicated to the former, fluids that are in motion, and how the fluid-flow behavior influences processes that may include heat transfer and possibly chemical reactions in combusting flows. This directly applies to the "*fluid dynamics*" description appearing in the terminology. Additionally, the physical characteristics of the fluid motion can usually be described through fundamental *mathematical* equations, usually in partial differential form, which govern a process of interest and are often called governing equations in CFD (see Chapter 3 for more insights). In order to solve these *mathematical* equations, *computer scientists* using high-level computer programming languages convert the equations into computer programs or software packages. The "*computational*" part simply means the study of the fluid flow

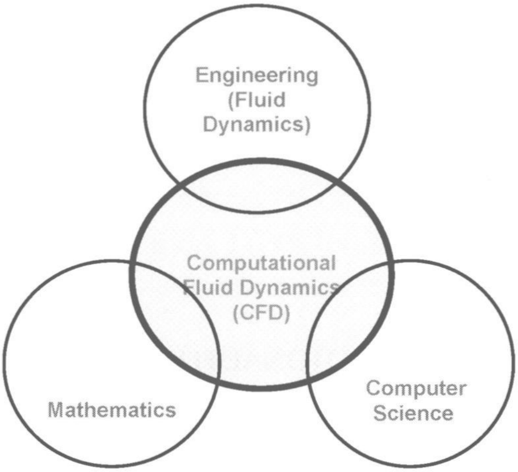


FIGURE 1.1 The different disciplines involved in computational fluid dynamics.

using numerical simulations, which involves employing computer programs or software packages performed on high-speed digital computers to attain the numerical solutions. Another question arises: Do we actually require the expertise of three specific people from each discipline—fluids engineering, mathematics, and computer science—to come together for the development of CFD programs or even to conduct CFD simulations? The answer is that it is more likely that a person who proficiently obtains more or less some subsets of the knowledge from each discipline can meet the demands of CFD.

CFD has also become one of the three basic methods or approaches that can be employed to solve problems in fluid dynamics and heat transfer. As demonstrated in Figure 1.2, the approaches that are strongly interlinked do not work in

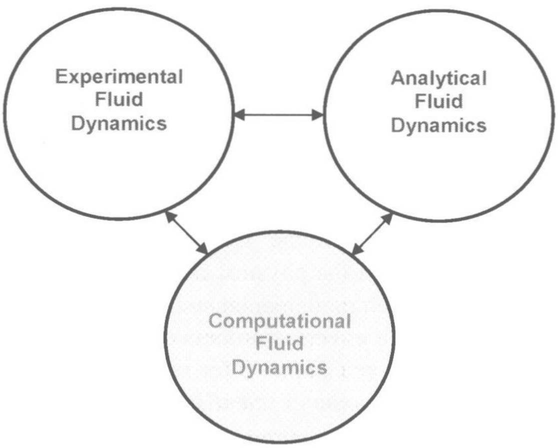


FIGURE 1.2 The three basic approaches to solve problems in fluid dynamics and heat transfer.