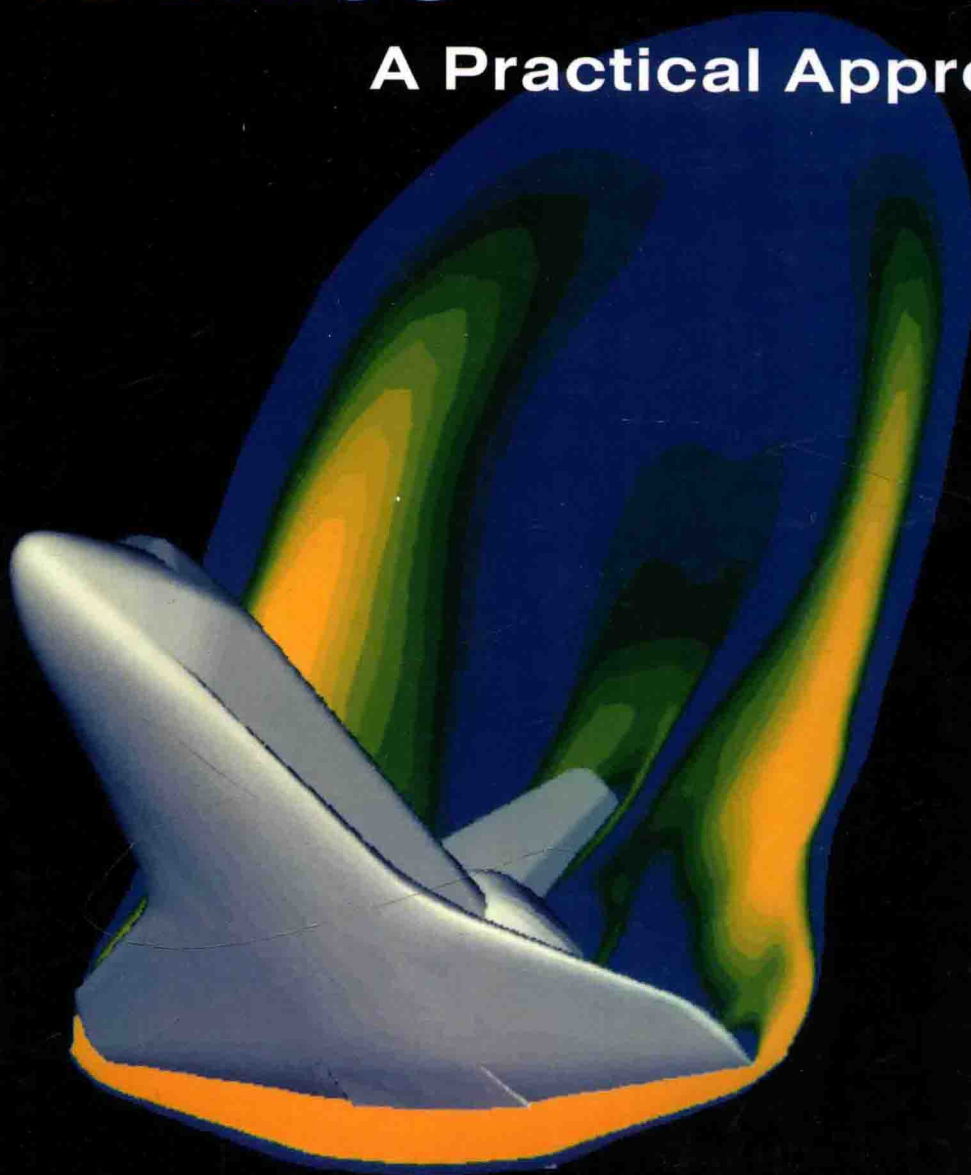


Jiyuan Tu Guan Heng Yeoh Chaoqun Liu

COMPUTATIONAL FLUID DYNAMICS

A Practical Approach



Computational Fluid Dynamics

A Practical Approach

Jiyuan Tu

RMIT University, Australia

Guan Heng Yeoh

Australian Nuclear Science and Technology Organisation

Chaoqun Liu

University of Texas, 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
30 Corporate Drive, Suite 400, Burlington, MA 01803, USA
Linacre House, Jordan Hill, Oxford OX2 8DP, UK

First edition 2008

Copyright © 2008, Elsevier Inc. All rights reserved.

No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise, without the prior written permission of the publisher.

Permissions may be sought directly from Elsevier's Science & Technology Rights Department in Oxford, UK: phone: (+44) 1865 843830, fax: (+44) 1865 853333, E-mail: permissions@elsevier.com. You may also complete your request online via the Elsevier homepage (<http://elsevier.com>), by selecting "Support & Contact" then "Copyright and Permission" and then "Obtaining Permissions."

ANSYS®, CFX®, FLUENT® and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license.

STAR-CD® is a registered trademark of CD-adapco.



Recognizing the importance of preserving what has been written, Elsevier prints its books on acid-free paper whenever possible.

Library of Congress Cataloging-in-Publication Data

Tu, Jiyuan.

Computational fluid dynamics: a practical approach/Jiyuan Tu, Guan Heng Yeoh, Chaoqun Liu.
p. cm.

Includes bibliographical references and index.
ISBN 978-0-7506-8563-4 (alk. paper)

1. Fluid dynamics. 2. Heat-Transmission. 3. Turbulence. I. Yeoh, Guan Heng. II. Liu, Chaoqun. III. Title.
TA357.T835 2007
620.1'06-dc22

British Library Cataloguing-in-Publication Data

A catalogue record for this book is available from the British Library.

ISBN: 978-0-7506-8563-4

For information on all Butterworth-Heinemann publications
visit our Web site at www.books.elsevier.com

Printed in the USA

07 08 09 10 11 12

10 9 8 7 6 5 4 3 2 1

Working together to grow
libraries in developing countries

www.elsevier.com | www.bookaid.org | www.sabre.org

ELSEVIER

BOOK AID
International

Sabre Foundation

Computational Fluid Dynamics

Dynamics

A Practical Approach

Enrique Tu

MIT / Prof. John A. Doolittle

Guang Heng Yeou

Aerospace Nuclear Science and Technology Organization

Chaochun Tu

University of Texas, Arls

Preface

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 towards the use of commercial CFD software from ANSYS Inc., (ANSYS[®], CFX[®], and 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 and application in CFD is acquired. Furthermore, changing workplace environments have 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 were conscientious of the limitations of CFD from their own experiences is fast dwindling.

The purpose of this book is to offer CFD users with a *suitable text* pitched at the *right level* of assumed knowledge. CFD is a mathematically sophisticated discipline and the authors' aim in this book is to provide simple to understand descriptions of fundamental CFD theories, basic CFD techniques, and practical guidelines. Our aim is not 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 that captures the reader's attention. The dominant feature of the present book is to maintain practicality in understanding CFD.

This book incorporates specially designed *intuitive* and *systematic* CFD examples that enhance the learning process and provide students with practice examples to better comprehend the basic principles. We hope that *beginners* will find this approach help them to focus more on the *engineering practice* of CFD techniques.

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 be guided through of these basic processes using any commercial, shareware, and in-house CFD software. More importantly, it serves as a guidepost for the reader to other chapters relating to fundamental knowledge of CFD.

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 through 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 the specific techniques to solve 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 to a range of engineering disciplines. Effort has been made in this chapter to offer the reader a range of important and 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 appetite of readers for more to come in the evolutionary use of CFD in any new emerging areas of science and engineering.

Chapter 8 provides an introduction to more advanced CFD techniques and provides a springboard to those going on to more advanced applications and courses.

Jiyuan Tu

Guan Heng Yeoh

Chaoqun Liu

Accompanying Resources

The following accompanying web-based resources are available for teachers and lecturers who adopt or recommend this text for class use. For further details and access to these resources please visit <http://www.textbooks.elsevier.com> and follow the registration instructions.

Instructor's Manual

A full set of worked solutions for the exercises in the main text are available for teaching purposes.

Image Bank

An image bank of downloadable figures from the book is available for use in lecture slides and class presentations.

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 of Technology (RMIT) University, Australia. For the students who have undertaken the course, we would like to thank them for providing us with feedback to better design particular project topics and aid understanding in this subject.

The authors would also like to thank many research students and colleagues who have generously assisted us in various ways towards the completion of this book. We would like to give special thanks to *Kiao Inthavong, Zhaofeng Tian, Krishna Mohanarangam, Weng Chan, Patrick Li, Gokul Rajamani, Huafeng Li, Sherman Cheung, Edsil Dilla* and *Mark Ho* for their skillful and meticulous preparation of the necessary questions, assignments, projects, and research in giving shape to this final version.

Mr. Jonathan Simpson, senior commissioning editor, and his colleagues at Elsevier have offered us immense help both in their academic elucidation and professional skills in the publication process.

Our special thanks are to *Dr. Risa Robinson*, Associate Professor of mechanical engineering at the Rochester Institute of Technology, USA, for reviewing the whole text within a very short period of time and giving us invaluable suggestions and comments which we have incorporated in this book.

Dr. Tu would like to express his deep gratitude to his wife, *Xue*, and his son, *Tian*, who have provided their unflinching support in preparing and writing the text.

Dr. Yeoh would also like to acknowledge the untiring support of his wife, *Natalie*, and his daughters, *Genevieve* and *Ellana*, for their understanding and encouragement during the seemingly unending hours spent in preparing and writing.

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

Contents

Preface	xi
Acknowledgments	xv

CHAPTER 1

Introduction	1
1.1 What is computational fluid dynamics	1
1.2 Advantages of computational fluid dynamics	4
1.3 Application of computational fluid dynamics	6
1.3.1 As a research tool	6
1.3.2 As an education tool to learn basic thermal-fluid science	8
1.3.3 As a design tool	9
1.3.4 Aerospace	9
1.3.5 Automotive engineering	10
1.3.6 Chemical and mineral processing	15
1.3.7 Civil and environmental engineering	18
1.3.8 Power generation	20
1.3.9 Sports	22
1.4 The future of computational fluid dynamics	24
1.5 Summary	26
Review questions	27

CHAPTER 2

CFD Solution Procedure—A Beginning	29
2.1 Introduction	29
2.2 Problem setup—pre-process	33
2.2.1 Creation of geometry—step 1	33
2.2.2 Mesh generation—step 2	35

2.2.3	Selection of physics and fluid properties—step 3	37
2.2.4	Specification of boundary conditions—step 4	40
2.3	Numerical solution—CFD solver	46
2.3.1	Initialization and solution control—step 5	46
2.3.2	Monitoring convergence—step 6	49
2.4	Result report and visualization—post-process	52
2.4.1	X-Y plots	53
2.4.2	Vector plots	55
2.4.3	Contour plots	57
2.4.4	Other plots	60
2.4.5	Data report and output	60
2.4.6	Animation	61
2.5	Summary	62
	Review questions	63

CHAPTER

3

Governing Equations for CFD—Fundamentals		65
3.1	Introduction	65
3.2	The continuity equation	65
3.2.1	Mass conservation	65
3.2.2	Physical interpretation	69
3.2.3	Comments	74
3.3	The momentum equation	75
3.3.1	Force balance	75
3.3.2	Physical interpretation	78
3.3.3	Comments	89
3.4	The energy equation	90
3.4.1	Energy conservation	90
3.4.2	Physical interpretation	93
3.4.3	Comments	100
3.5	The additional equations for turbulent flow	100
3.5.1	What is turbulence	100
3.5.2	k - ϵ Two-equation turbulence model	102
3.5.3	Comments	109

3.6	Generic form of the governing equations for CFD	110
3.7	Physical boundary conditions of the governing equations	117
3.8	Summary	120
	Review questions	122

CHAPTER 4

	CFD Techniques—The Basics	126
4.1	Introduction	126
4.2	Discretization of governing equations	128
4.2.1	Finite-difference method	128
4.2.2	Finite-volume method	134
4.2.3	Converting governing equations to algebraic equation system	139
4.3	Numerical solutions to algebraic equations	152
4.3.1	Direct methods	153
4.3.2	Iterative methods	157
4.3.3	Pressure-velocity coupling—SIMPLE scheme	163
4.4	Summary	175
	Review questions	176

CHAPTER 5

	CFD Solution Analysis—Essentials	180
5.1	Introduction	180
5.2	Consistency	181
5.3	Stability	185
5.4	Convergence	192
5.4.1	What is convergence	192
5.4.2	Residuals and convergence tolerance	193
5.4.3	Convergence difficulty and using under-relaxation	196
5.4.4	Accelerating convergence	198
5.5	Accuracy	199
5.5.1	Source of solution errors	200
5.5.2	Controlling the solution errors	206
5.5.3	Verification and validation	208

5.6	Efficiency	209
5.7	Case studies	212
5.7.1	Test case A: channel flow	212
5.7.2	Test case B: flow over a 90° bend	215
5.8	Summary	221
	Review questions	222

CHAPTER**6****Practical Guidelines for CFD Simulation and Analysis** 224

6.1	Introduction	224
6.2	Guidelines on grid generation	225
6.2.1	Overview of grid generation	225
6.2.2	Guidelines on grid quality and grid design	234
6.2.3	Local refinement and solution adaptation	238
6.3	Guidelines on boundary conditions	242
6.3.1	Overview of setting boundary conditions	242
6.3.2	Guidelines on inlet boundary conditions	244
6.3.3	Guidelines on outlet boundary conditions	246
6.3.4	Guidelines on wall boundary conditions	247
6.3.5	Guidelines on symmetry and periodic boundary conditions	248
6.4	Guidelines on turbulence modeling	250
6.4.1	Overview of turbulence modeling approaches	250
6.4.2	Strategy for selecting turbulence models	253
6.4.3	Near-wall treatments	259
6.4.4	Setting boundary conditions	263
6.4.5	Test case: assessment of two-equation turbulence modeling for hydrofoil flows	265
6.5	Summary	271
	Review questions	272

CHAPTER**7****Some Applications of CFD with Examples** 277

7.1	Introduction	277
7.2	To assist in design process—as a design tool	278

7.2.1	Indoor airflow distribution	278
7.3	To enhance understanding—as a research tool	286
7.4	Other important applications	295
7.4.1	Heat transfer coupled with fluid flow	295
7.4.2	A buoyant free-standing fire	314
7.4.3	Flow over vehicle platoon	320
7.4.4	Air/Particle flow in the human nasal cavity	326
7.4.5	High-speed flows	334
7.5	Summary	356
	Review questions	358

CHAPTER 8

	Some Advanced Topics in CFD	364
8.1	Introduction	364
8.2	Advances in numerical methods and techniques	364
8.2.1	Incompressible flows	364
8.2.2	Compressible flows	368
8.2.3	Moving grids	375
8.2.4	Multigrid methods	377
8.2.5	Parallel computing	379
8.2.6	Immersed boundary methods	381
8.3	Advances in computational models	384
8.3.1	Direct numerical simulation (DNS)	384
8.3.2	Large eddy simulation (LES)	387
8.3.3	RANS-LES coupling for turbulent flows	390
8.3.4	Multiphase flows	392
8.3.5	Combustion	394
8.3.6	Fluid-structure interaction	396
8.3.7	Physiological fluid dynamics	398
8.4	Other numerical approaches for computation of fluid dynamics	400
8.4.1	Lattice Boltzmann method	401
8.4.2	Monte Carlo method	402
8.4.3	Particle methods	404

8.5 Summary	406
Review questions	407

APPENDIX**A**

Full Derivation of Conservation Equations	410
---	-----

APPENDIX**B**

Upwind Schemes	414
----------------	-----

APPENDIX**C**

Explicit and Implicit Methods	416
-------------------------------	-----

APPENDIX**D**

Learning Program	418
------------------	-----

APPENDIX**E**

CFD Assignments and Guideline for CFD Project	421
---	-----

References	437
------------	-----

Further Suggested Reading	453
---------------------------	-----

Index	455
-------	-----

Chapter 1 / Introduction

1.1 WHAT IS COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics has certainly come of age in industrial applications and academia research. In the beginning this popular field of study was primarily limited to high-technology engineering areas of aeronautics and astronautics, but now it is a widely adopted methodology for solving complex problems in many modern engineering fields. CFD, derived from different disciplines of fluid mechanics and heat transfer, is also finding its way into other important uncharted areas especially in process, chemical, civil, and environmental engineering. Construction of new and better improved system designs and optimization carried out on existing equipments through computational simulations are resulting in enhanced efficiency and lower operating costs. With the concerns of global warming and increasing world 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 meeting strict regulation challenges of emissions control and substantial reduction of environmental pollutants.

Nevertheless, the basic question remains: What actually is *computational fluid dynamics*? In retrospect, it has certainly become a new branch integrating not only the disciplines of fluid mechanics with mathematics but also with computer science as illustrated in Fig. 1.1. Let us 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 combust-ing flows. This directly infers 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

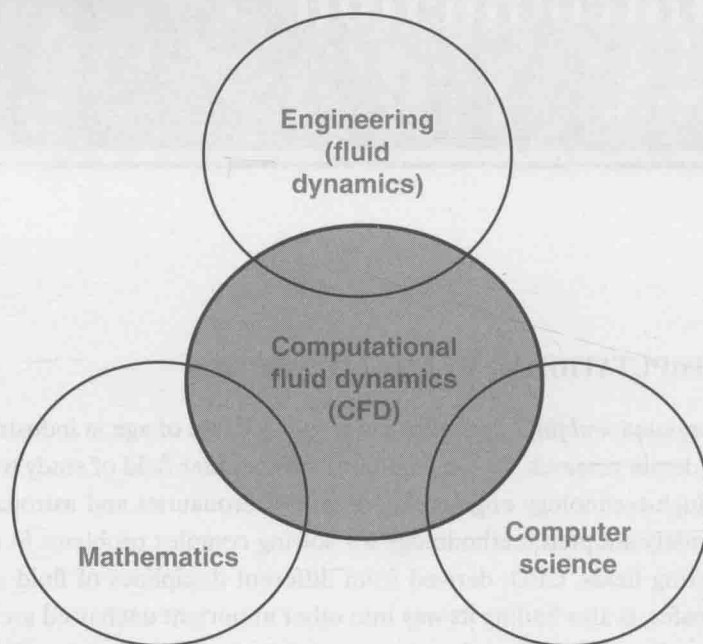


FIGURE 1.1 The different disciplines contained within computational fluid dynamics

mathematical equations, they are converted by *computer scientists* using high-level computer programming languages into computer programs or software packages. The *computational* part simply means the study of the fluid flow through 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 obviously no, and more likely it is expected that this field demands a person who will proficiently obtain some subsets of the knowledge from each discipline.

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 Fig. 1.2, each approach is strongly interlinked and does not lie in isolation. Traditionally, both experimental and analytical methods have been used to study the various aspects of fluid dynamics and to assist engineers in the design of equipment and industrial processes involving fluid flow and heat transfer. With the advent of digital computers, the computational (numerical) aspect has emerged as another viable approach. Although the analytical method is still practiced by many and experiments will continue to be significantly performed, the trend is clearly toward greater reliance

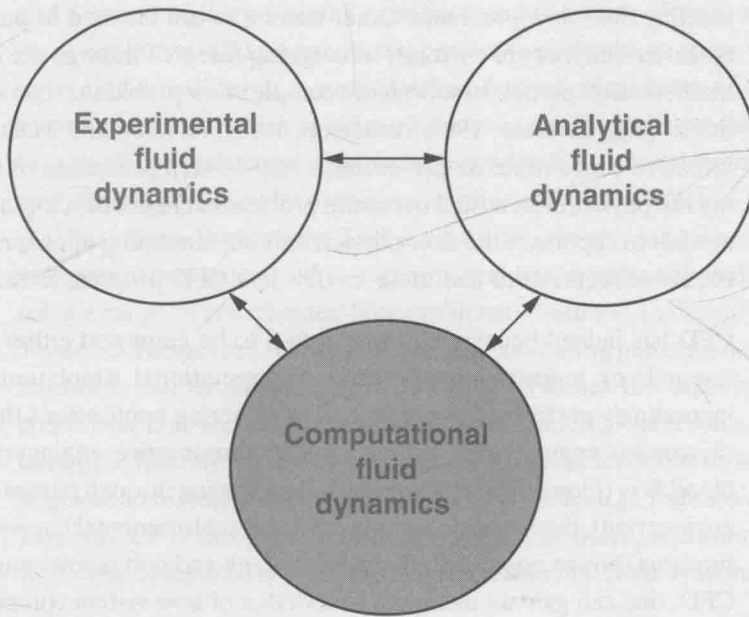


FIGURE 1.2 The three basic approaches to solve problems in fluid dynamics and heat transfer

on the computational approach for industrial designs, particularly when the fluid flows are very complex.

In the past, potential or novice users would probably learn CFD by investing a substantial amount of time writing their own computer programs. With the increasing demands from industries or even within academia to acquire the knowledge of CFD in a much shorter time frame, it is not surprising that the interest in abandoning writing computer programs is escalating in favor of using more commercially available software packages. Multipurpose CFD programs are gradually earning the approval, and with the advancement of models that better encapsulate the flow physics, these software packages are also gaining wide acceptance. There are numerous advantages in applying these computer programs. Since the mundane groundwork of writing and testing of these computer codes has been thoroughly carried out by the “developers” of respective software companies, today’s potential or novice CFD users are comforted by not having to deal with these types of issues. Such a program can be readily employed to solve numerous fluid-flow problems.

Despite the well-developed methodologies within the computational codes, CFD is certainly more than just being proficient in operating these software packages. Bearing this in mind, the primary focus of this book is thus oriented to better educate potential or novice users in employing CFD in a more judicious manner, equally supplementing the understanding of underlying basic concepts and the technical know-how in better