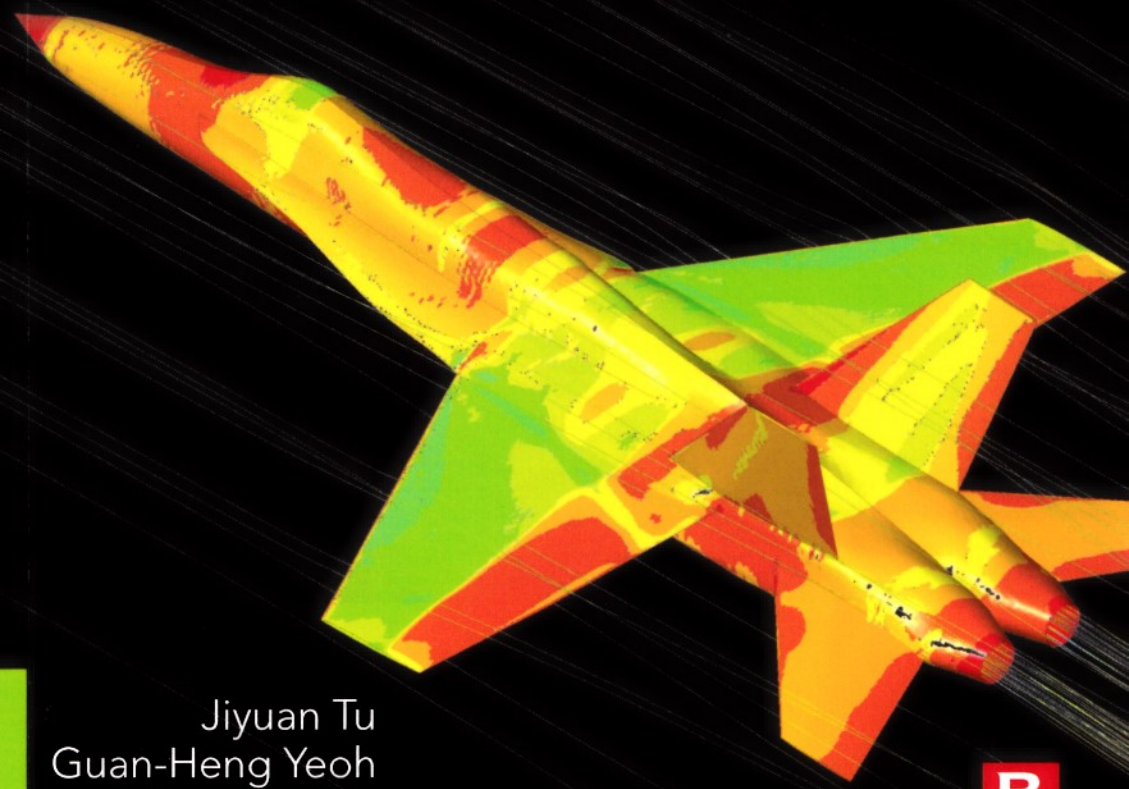
Computational Fluid Dynamics

THIRD EDITION

A Practical Approach



Jiyuan Tu
Guan-Heng Yeoh
Chaoqun Liu

B
H

Contents*

Preface for Third Edition xi
Preface for Second Edition xiii
Preface for First Edition xv
Acknowledgements xix

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 Educational Tool 7
1.3.3 As a Design Tool 8
1.3.4 Aerospace 9
1.3.5 Automotive Engineering 10
1.3.6 Biomedical Science and Engineering 12
1.3.7 Chemical and Mineral Processing 15
1.3.8 Civil and Environmental Engineering 17
1.3.9 Metallurgy 19
1.3.10 Nuclear Safety 20
1.3.11 Power Generation and Renewable Energy 22
1.3.12 Sports 25
1.4 The Future of Computational Fluid Dynamics 27
1.5 Summary 29
Review Questions 30

2. CFD Solution Procedure: A Beginning 33

2.1 Introduction 33
2.1.1 Shareware CFD 34
2.1.2 Commercial CFD 35
2.2 Problem Setup—Preprocess 36
2.2.1 Creation of Geometry—Step 1 36
2.2.2 Mesh Generation—Step 2 39
2.2.3 Selection of Physics and Fluid Properties—Step 3 40
2.2.4 Specification of Boundary Conditions—Step 4 42

*For instructors using this book, available teaching resources can be obtained online by registering on www.textbooks.elsevier.com.

2.3 Numerical Solution—CFD Solver 46
2.3.1 Initialization and Solution Control—Step 5 46
2.3.2 Monitoring Convergence—Step 6 48
2.4 Result Report and Visualization—Postprocess 51
2.4.1 *X–Y* Plots 53
2.4.2 Vector Plots 54
2.4.3 Contour Plots 56
2.4.4 Other Plots 58
2.4.5 Data Report and Output 59
2.4.6 Animation 60
2.5 Summary 61
Review Questions 62

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 72
3.3 The Momentum Equation 73
3.3.1 Force Balance 73
3.3.2 Physical Interpretation 77
3.3.3 Comments 87
3.4 The Energy Equation 88
3.4.1 Energy Conservation 88
3.4.2 Physical Interpretation 91
3.4.3 Comments 97
3.5 The Additional Equations for Turbulent Flow 98
3.5.1 What is Turbulence 98
3.5.2 k-ε Two-Equation Turbulence Model 100
3.5.3 Comments 108
3.6 Generic Form of the Governing Equations for CFD 108
3.7 Physical Boundary Conditions of the Governing Equations 117
3.8 Summary 120
Review Questions 121

4. CFD Mesh Generation: A Practical Guideline 125

4.1 Introduction 125
4.2 Types of Meshes 127
4.2.1 Structured Mesh 127
4.2.2 Body-Fitted Mesh 128
4.2.3 Unstructured Mesh 133
4.3 Comments on Mesh Topology 136
4.4 Local Mesh Refinement 140
4.5 Overlapping Mesh Techniques 144
4.6 Adaptive Mesh With Solution 145
4.7 Moving Meshes 146

4.8 Guidelines for Mesh Quality and Mesh Design 148
4.9 Summary 152
Review Questions 152

5. CFD Techniques: The Basics 155

5.1 Introduction 155
5.2 Discretization of Governing Equations 158
5.2.1 Finite Difference Method 158
5.2.2 Finite Volume Method 163
5.2.3 Finite Element Method 168
5.2.4 Spectral Method 169
5.3 Converting Governing Equations to Algebraic Equation System 169
5.3.1 Finite Difference Method 170
5.3.2 Finite Volume Method 171
5.3.3 Comparison of the Finite Difference and Finite Volume Discretizations 173
5.4 Numerical Solutions to Algebraic Equations 184
5.4.1 Direct Methods 185
5.4.2 Iterative Methods 189
5.5 Pressure–Velocity Coupling—Simple Scheme 193
5.6 Multigrid Method 204
5.7 Summary 206
Review Questions 207

6. CFD Solution Analysis: Essentials 211

6.1 Introduction 211
6.2 Consistency 212
6.3 Stability 216
6.4 Convergence 222
6.4.1 What is Convergence 222
6.4.2 Residuals and Convergence Tolerance 224
6.4.3 Convergence Difficulty and Using Under-Relaxation 227
6.4.4 Accelerating Convergence 228
6.5 Accuracy 229
6.5.1 Source of Solution Errors 230
6.5.2 Controlling the Solution Errors 236
6.5.3 Verification and Validation 238
6.6 Efficiency 239
6.7 Case Studies 241
6.7.1 Test Case A: Channel Flow 242
6.7.2 Test Case B: Flow over a 90° Bend 245
6.8 Summary 250
Review Questions 251

7. Practical Guidelines for CFD Simulation and Analysis 255

7.1 Introduction 255
7.2 Guidelines for Boundary Conditions 256
7.2.1 Overview of Setting Boundary Conditions 256

7.2.2 Guidelines for Inlet Boundary Conditions 258
7.2.3 Guidelines for Outlet Boundary Conditions 260
7.2.4 Guidelines for Wall Boundary Conditions 261
7.2.5 Guidelines on Symmetry and Periodic Boundary Conditions 262
7.3 Guidelines for Turbulence Modeling 264
7.3.1 Overview of Turbulence Modeling Approaches 264
7.3.2 Strategy for Selecting Turbulence Models 267
7.3.3 Near Wall Treatments 273
7.3.4 Setting Boundary Conditions 277
7.3.5 Test Case: Assessment of Two-Equation Turbulence Modeling for Hydrofoil Flows 279
7.4 Other Practical Guidelines for CFD Users 284
7.4.1 Guidelines on Problem Definition 284
7.4.2 Guidelines on Solution Strategy and Global Solution Algorithm 285
7.4.3 Guidelines on Validation 286
7.5 Summary 286
Review Questions 288

8. Some Applications of CFD With Examples 291

8.1 Introduction 291
8.2 To Assist in Design Process—As a Design Tool 292
8.2.1 Indoor Airflow Distribution 292
8.3 To Enhance Understanding—As a Research Tool 298
8.3.1 Gas-Particle Flow in a 90 degrees Bend 299
8.4 Other Important Applications 305
8.4.1 Heat Transfer Coupled With Fluid Flow 305
8.4.2 A Buoyant Free-Standing Fire 320
8.4.3 Flow Over Vehicle Platoon 325
8.4.4 Air/Particle Flow in the Human Nasal Cavity 331
8.4.5 High Speed Flows 338
8.5 Summary 361
Review Questions 362

9. Some Advanced Topics in CFD 369

9.1 Introduction 369
9.2 Advances in Numerical Methods and Techniques 369
9.2.1 Incompressible Flows 369
9.2.2 Compressible Flows 373
9.2.3 Moving Grids 380
9.2.4 Multigrid Methods 381
9.2.5 Parallel Computing 384
9.2.6 Graphics Processing Units and CUDA Environment 385
9.2.7 Immersed Boundary Methods 387
9.3 Advances in Computational Models 390
9.3.1 Direct Numerical Simulation (DNS) 390

9.3.2 Large Eddy Simulation (LES) 393
9.3.3 RANS–LES Coupling for Turbulent Flows 396
9.3.4 Multiphase Flows 398
9.3.5 Combustion 400
9.3.6 Fluid–Structure Interaction 402
9.3.7 Physiological Fluid Dynamics 404
9.4 Other Numerical Approaches for Computation of Fluid Dynamics 406
9.4.1 Lattice Boltzmann Method 407
9.4.2 Monte Carlo Method 409
9.4.3 Particle Methods 411
9.4.4 Discrete Element Method 413
9.5 Summary 415
Review Questions 416

e1. CFD Case Studies https://www.elsevier.com/books-and-journals/book-companion/9780081011270 e1

Appendix A: Full Derivation of Conservation Equations 419
Appendix B: Upwind Schemes 427
Appendix C: Explicit and Implicit Methods 429
Appendix D: Learning Program 431
Appendix E: CFD Assignments and Guideline for CFD Project 435
References 451

Index 467

Additional material available online.

THIRD EDITION

Computational Fluid Dynamics
A Practical Approach

Jiyuan Tu, Guan-Heng Yeoh, and Chaoqun Liu

An introduction to understanding CFD and using CFD codes to solve engineering problems, designed for the CFD user rather than CFD developer.

Computational Fluid Dynamics: A Practical Approach is an introduction to CFD fundamentals and commercial CFD software to solve engineering problems, designed for the wide variety of engineering students new to CFD and for practicing engineers learning CFD for the first time. Combining an appropriate level of mathematical background, worked examples, computer screenshots, and step-by-step processes, this book walks the reader through modelling and computing, as well as interpreting CFD results. This new edition has been updated throughout, with new content and improved figures, examples, and problems.

Key Features:
- Includes a new chapter on practical guidelines for mesh generation.
- Provides full coverage of high-pressure fluid dynamics and meshless approach to provide a broader overview of the application areas where CFD can be used.
- Online resources include a new bonus chapter featuring detailed case studies and the latest developments in CFD.

Related Titles

Blazek / Computational Fluid Dynamics: Principles and Applications, Third Edition / 9780080999951

Hirsch / Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics, Second Edition / 9780750665940

Yeoh and Yuen / Computational Fluid Dynamics in Fire Engineering / 9780750685894

About the Authors

Jiyuan Tu, Distinguished Professor, Tsinghua University, P.R.China, Adjunct Professor, University of NSW, Australia, and Professor, RMIT University, Australia

Guan-Heng Yeoh, Principal Research Scientist, ANSTO (Australian Nuclear Science and Technology Organization), Professor, University of NSW, Australia, and Visiting Professor at the Zhejiang University and City University of Hong Kong

Chaoqun Liu, Director of the Center for Numerical Simulation and Modeling, and Distinguished Professor, University of Texas at Arlington, USA

Engineering, Fluid Dynamics

ISBN 978-0-08-101127-0

9 780081 011270

ELSEVIER

Butterworth-Heinemann
An imprint of Elsevier
elsevier.com/books-and-journals