Engineering Applications Of Computational Fluid Mechanics

Computational Fluid DynamicsComputational Fluid Dynamics: Principles and ApplicationsEngineering Applications of Unsteady Fluid FlowNumerical SimulationComputational Fluid DynamicsEngineering Applications of Computational Fluid DynamicsComputational Fluid Dynamics with Moving BoundariesComputational Fluid DynamicsApplied Computational Fluid DynamicsComputational Fluid Dynamics for Engineers and ScientistsMultiphysics Modeling: Numerical Methods and Engineering ApplicationsComputational Fluid Dynamics: Principles and ApplicationsFundamentals of Computational Fluid DynamicsComputational Fluid Dynamics (CFD)Lattice Boltzmann Method And Its Application In EngineeringComputational Fluid DynamicsThe Finite Volume Method in Computational Fluid DynamicsComputational Fluid Dynamics Applications in Food ProcessingEssentials of Computational Fluid DynamicsOptimization and Computational Fluid DynamicsComputational Fluid Dynamics Applied to Waste-to-Energy ProcessesHandbook of Computational Fluid MechanicsComputational Methods in Environmental Fluid MechanicsComputational Fluid Dynamics and Heat TransferComputational Fluid Dynamics in Food ProcessingComputational Fluid DynamicsUsing HPC for Computational Fluid DynamicsApplied and Computational Fluid MechanicsIntroduction to Computational Fluid DynamicsComputational Fluid Dynamics in Industrial CombustionComputational Fluid Dynamics Simulation of Spray DryersComputational Fluid Dynamics for EngineersEnvironmental ModellingAdvances of Computational Fluid Dynamics in Nuclear Reactor Design and Safety AssessmentComputational Fluid DynamicsApplied Computational Fluid Dynamics and Turbulence ModelingComputational Fluid-Structure InteractionComputational Fluid-Structure InteractionComputational Fluid DynamicsComputational Fluid Dynamics in Fire Engineering

Computational Fluid Dynamics

Includes bibliographical references and index.

Computational Fluid Dynamics: Principles and Applications

Designed for the fluid mechanics course for mechanical, civil, and aerospace engineering students, or as a reference for professional engineers, this up to date text uses computer algorithms and applications to solve modern problems related to fluid flow, aerodynamics, and thermodynamics. Algorithms and codes for numerical solutions of fluid problems, which can be implemented in programming environments such as MATLAB, are used throughout the book. The author also uses non-language specific algorithms to force the students to think through the logic of the solution technique as they translate the algorithm into the software they are using. The text also includes an introduction to Computational Fluid Dynamics, a well-established method in the design of fluid machinery and heat transfer applications. A DVD accompanies every new printed copy of the book and contains the source code, MATLAB files, third-party simulations, color figures, and more.

Engineering Applications of Unsteady Fluid Flow

Heat transfer and fluid flow issues are of great significance and this state-of-the-art edited book with reference to new and innovative numerical methods will make a contribution for researchers in academia and research organizations, as well as industrial scientists and college students. The book provides comprehensive chapters on research and developments in emerging topics in computational methods, e.g., the finite volume method, finite element method as well as turbulent flow computational methods. Fundamentals of the numerical methods, comparison of various higher-order schemes for convection-diffusion terms, turbulence modeling, the pressure-velocity coupling, mesh generation and the handling of arbitrary geometries are presented. Results from engineering applications are provided. Chapters have been co-authored by eminent researchers.

Numerical Simulation

Computational Fluid Dynamics Applied to Waste-to-Energy Processes: A Hands-On Approach provides the key knowledge needed to perform CFD simulations using powerful commercial software tools. The book focuses on fluid mechanics, heat transfer and chemical reactions. To do so, the fundamentals of CFD are presented, with the entire workflow broken into manageable pieces that detail geometry preparation, meshing, problem setting, model implementation and post-processing actions. Pathways for process optimization using CFD integrated with Design of Experiments are also explored. The book's combined approach of theory, application and hands-on practice allows engineering graduate students, advanced undergraduates and industry practitioners to develop their own simulations. Provides the skills needed to perform real-life simulation calculations through a combination of mathematical background and real-world examples, including step-by-step tutorials Presents worked examples in complex processes as combustion or gasification involving fluid dynamics, heat and mass transfer, and complex chemistry sets

Computational Fluid Dynamics

The chosen semi-discrete approach of a reduction procedure of partial differential equations to ordinary differential equations and finally to difference equations gives the book its distinctiveness and provides a sound basis for a deep understanding of the fundamental concepts in computational fluid dynamics.

Engineering Applications of Computational Fluid Dynamics

"Describes the latest techniques and real-life applications of computational fluid dynamics (CFD) and heat transfer in aeronautics, materials processing and manufacturing, electronic cooling, and environmental control. Includes new material from experienced researchers in the field. Complete with detailed equations for fluid flow and heat transfer.

Computational Fluid Dynamics with Moving Boundaries

Bridging the gap in understanding between the spray drying industry and the numerical modeler on spray drying, Computational Fluid Dynamics Simulation of Spray Dryers: An Engineer's Guide shows how to numerically capture important physical phenomena within a spray drying process using the CFD technique. It includes numerical strategies to effectively describe these phenomena, which are collated from research work and CFD industrial consultation, in particular to the dairy industry. Along with showing how to set up models, the book helps readers identify the capabilities and uncertainties of the CFD technique for spray drying. After briefly covering the basics of CFD, the book discusses airflow modeling, atomization and particle tracking, droplet drying, quality modeling, agglomeration and wall deposition modeling, and simulation validation techniques. The book also answers questions related to common challenges in industrial applications.

Computational Fluid Dynamics

Nowadays mathematical modeling and numerical simulations play an important role in life and natural science. Numerous researchers are working in developing different methods and techniques to help understand the behavior of very complex systems, from the brain activity with real importance in medicine to the turbulent flows with important applications in physics and engineering. This book presents an overview of some models, methods, and numerical computations that are useful for the applied research scientists and mathematicians, fluid tech engineers, and postgraduate students.

Applied Computational Fluid Dynamics

Lattice Boltzmann method (LBM) is a relatively new simulation technique for the modeling of complex fluid systems and has attracted interest from researchers in computational physics. Unlike the traditional CFD methods, which solve the conservation equations of macroscopic properties (i.e., mass, momentum, and energy) numerically, LBM models the fluid consisting of fictive particles, and such particles perform consecutive propagation and collision processes over a discrete lattice mesh.This book will cover the fundamental and practical application of LBM. The first part of the book consists of three chapters starting form the theory of LBM, basic models, initial and boundary conditions, theoretical analysis, to improved models. The second part of the book consists of six chapters, address applications of LBM in various aspects of computational fluid dynamic engineering, covering areas, such as thermo-hydrodynamics, compressible flows, multicomponent/multiphase flows, microscale flows, flows in porous media, turbulent flows, and suspensions.With these coverage LBM, the book intended to promote its applications, instead of the traditional computational fluid dynamic method.

Computational Fluid Dynamics for Engineers and Scientists

Advances of Computational Fluid Dynamics in Nuclear Reactor Design and Safety Assessment presents the latest computational fluid dynamic technologies. It includes an evaluation of safety systems for reactors using CFD and their design, the modeling of Severe Accident Phenomena Using CFD, Model Development for

Two-phase Flows, and Applications for Sodium and Molten Salt Reactor Designs. Editors Joshi and Nayak have an invaluable wealth of experience that enables them to comment on the development of CFD models, the technologies currently in practice, and the future of CFD in nuclear reactors. Readers will find a thematic discussion on each aspect of CFD applications for the design and safety assessment of Gen II to Gen IV reactor concepts that will help them develop cost reduction strategies for nuclear power plants. Presents a thematic and comprehensive discussion on each aspect of CFD applications for the design and safety assessment of nuclear reactors Provides an historical review of the development of CFD models, discusses state-of-the-art concepts, and takes an applied and analytic look toward the future Includes CFD tools and simulations to advise and guide the reader through enhancing cost effectiveness, safety and performance optimization

Multiphysics Modeling: Numerical Methods and Engineering Applications

Computational fluid dynamics, CFD, has become an indispensable tool for many engineers. This book gives an introduction to CFD simulations of turbulence, mixing, reaction, combustion and multiphase flows. The emphasis on understanding the physics of these flows helps the engineer to select appropriate models to obtain reliable simulations. Besides presenting the equations involved, the basics and limitations of the models are explained and discussed. The book combined with tutorials, project and power-point lecture notes (all available for download) forms a complete course. The reader is given hands-on experience of drawing, meshing and simulation. The tutorials cover flow and reactions inside a porous catalyst, combustion in turbulent non-premixed flow, and multiphase simulation of evaporation spray respectively. The project deals with design of an industrial-scale selective catalytic reduction process and allows the reader to explore various design improvements and apply best practice guidelines in the CFD simulations.

Computational Fluid Dynamics: Principles and Applications

Computational Fluid Dynamics (CFD) has been applied extensively to great benefit in the food processing sector. Its numerous applications include: predicting the gas flow pattern and particle histories, such as temperature, velocity, residence time, and impact position during spray drying; modeling of ovens to provide information about temperature and airflow pattern throughout the baking chamber to enhance heat transfer and in turn final product quality; designing hybrid heating ovens, such as microwave-infrared, infrared-electrical or microwave-electrical ovens for rapid baking; model the dynamics of gastrointestinal contents during digestion based on the motor response of the GI tract and the physicochemical properties of luminal contents; retort processing of canned solid and liquid foods for understanding and optimization of the heat transfer processes. This Brief will recapitulate the various applications of CFD modeling, discuss the recent developments in this field, and identify the strengths and weaknesses of CFD when applied in the food industry.

Fundamentals of Computational Fluid Dynamics

Using HPC for Computational Fluid Dynamics: A Guide to High Performance Computing for CFD Engineers offers one of the first self-contained guides on the use of high performance computing for computational work in fluid dynamics. Beginning with an introduction to HPC, including its history and basic terminology, the book moves on to consider how modern supercomputers can be used to solve common CFD challenges, including the resolution of high density grids and dealing with the large file sizes generated when using commercial codes. Written to help early career engineers and post-graduate students compete in the fast-paced computational field where knowledge of CFD alone is no longer sufficient, the text provides a one-stop resource for all the technical information readers will need for successful HPC computation. Offers one of the first self-contained guides on the use of high performance computing for computational work in fluid dynamics Tailored to the needs of engineers seeking to run CFD computations in a HPC environment

Computational Fluid Dynamics (CFD)

This handbook covers computational fluid dynamics from fundamentals to applications. This text provides a well documented critical survey of numerical methods for fluid mechanics, and gives a state-of-the-art description of computational fluid mechanics, considering numerical analysis, computer technology, and visualization tools. The chapters in this book are invaluable tools for reaching a deeper understanding of the problems associated with the calculation of fluid motion in various situations: inviscid and viscous, incompressible and compressible, steady and unsteady, laminar and turbulent flows, as well as simple and complex geometries. Each chapter includes a related bibliography Covers fundamentals and applications Provides a deeper understanding of the problems associated with the calculation of fluid motion

Lattice Boltzmann Method And Its Application In Engineering

Computational Fluid Dynamics enables engineers to model and predict fluid flow in powerful, visually impressive ways and is one of the core engineering design tools, essential to the study and future work of many engineers. This textbook is designed to explcitly meet the needs engineering students taking a first course in CFD or computer-aided engineering. Fully course matched, with the most extensive and rigorous pedagogy and features of any book in the field, it is certain to be a key text. The only course text available specifically designed to give an applications-lead, commercial software oriented approach to understanding and using Computational Fluid Dynamics (CFD). Meets the needs of all engineering disciplines that use CFD. The perfect CFD teaching resource: clear, straightforward text, step-by-step explanation of mathematical foundations, detailed worked examples, end-of-chapter knowledge check exercises, and homework assignment questions

Computational Fluid Dynamics

This volume presents the results of Computational Fluid Dynamics (CFD) analysis that can be used for conceptual studies of product design, detail product development, process troubleshooting. It demonstrates the benefit of CFD modeling as a cost saving, timely, safe and easy to scale-up methodology.

The Finite Volume Method in Computational Fluid Dynamics

This unique text provides engineering students and practicing professionals with a comprehensive set of practical, hands-on guidelines and dozens of step-by-step examples for performing state-of-the-art, reliable computational fluid dynamics (CFD) and turbulence modeling. Key CFD and turbulence programs are included as well. The text first reviews basic CFD theory, and then details advanced applied theories for estimating turbulence, including new algorithms created by the author. The book gives practical advice on selecting appropriate turbulence models and presents best CFD practices for modeling and generating reliable simulations. The author gathered and developed the book's hundreds of tips, tricks, and examples over three decades of research and development at three national laboratories and at the University of New Mexico—many in print for the first time in this book. The book also places a strong emphasis on recent CFD and turbulence advancements found in the literature over the past five to 10 years. Readers can apply the author's advice and insights whether using commercial or national laboratory software such as ANSYS Fluent, STAR-CCM, COMSOL, Flownex, SimScale, OpenFOAM, Fuego, KIVA, BIGHORN, or their own computational tools. Applied Computational Fluid Dynamics and Turbulence Modeling is a practical, complementary companion for academic CFD textbooks and senior project courses in mechanical, civil, chemical, and nuclear engineering; senior undergraduate and graduate CFD and turbulence modeling courses; and for professionals developing commercial and research applications.

Computational Fluid Dynamics Applications in Food Processing

Due to recent and continuing advances in computational fluid dynamics, programs are now available for speedy and detailed treatment of problems involving nonstationary fluid dynamics. Explains the basic ideas underlying the use of unsteady fluid flow and explores new ideas, processes and devices taking place in this rapidly developing field. Covers such diverse topics as tidal wave power, wind energy conversion systems, and thrust-augmenting pulsed ejectors. Offers numerous illustrated examples and applications to both inspire and challenge the reader.

Essentials of Computational Fluid Dynamics

Multiphysics Modeling: Numerical Methods and Engineering Applications: Tsinghua University Press Computational Mechanics Series describes the basic principles and methods for multiphysics modeling, covering related areas of physics such as structure mechanics, fluid dynamics, heat transfer, electromagnetic field, and noise. The book provides the latest information on basic numerical methods, also considering coupled problems spanning fluid-solid interaction, thermal-stress coupling, fluid-solid-thermal coupling, electromagnetic solid thermal fluid coupling,

and structure-noise coupling. Users will find a comprehensive book that covers background theory, algorithms, key technologies, and applications for each coupling method. Presents a wealth of multiphysics modeling methods, issues, and worked examples in a single volume Provides a go-to resource for coupling and multiphysics problems Covers the multiphysics details not touched upon in broader numerical methods references, including load transfer between physics, element level strong coupling, and interface strong coupling, amongst others Discusses practical applications throughout and tackles real-life multiphysics problems across areas such as automotive, aerospace, and biomedical engineering

Optimization and Computational Fluid Dynamics

The numerical optimization of practical applications has been an issue of major importance for the last 10 years. It allows us to explore reliable non-trivial configurations, differing widely from all known solutions. The purpose of this book is to introduce the state-of-the-art concerning this issue and many complementary applications are presented.

Computational Fluid Dynamics Applied to Waste-to-Energy Processes

Uniquely outlines CFD theory in a manner relevant to environmental applications. This book addresses the basic topics in CFD modelling in a thematic manner to provided the necessary theoretical background, as well as providing global cases studies showing how CFD models can be used in practice demonstrating how good practice can be achieved , with reference to both established and new applications. First book to apply CFD to the environmental sciences Written at a level suitable for non-mathematicians

Handbook of Computational Fluid Mechanics

This book is the result of a careful selection of contributors in the field of CFD. It is divided into three sections according to the purpose and approaches used in the development of the contributions. The first section describes the "high-performance computing" (HPC) tools and their impact on CFD modeling. The second section is dedicated to "CFD models for local and large-scale industrial phenomena." Two types of approaches are basically contained here: one concerns the adaptation from global to local scale, - e.g., the applications of CFD to study the climate changes and the adaptations to local scale. The second approach, very challenging, is the multiscale analysis. The third section is devoted to "CFD in numerical modeling approach for experimental cases." Its chapters emphasize on the numerical approach of the mathematical models associated to few experimental (industrial) cases. Here, the impact and the importance of the mathematical modeling in CFD are focused on. It is expected that the collection of these chapters will enrich the state of the art in the CFD domain and its applications in a lot of fields. This collection proves that CFD is a highly interdisciplinary research area, which lies at the interface of physics, engineering, applied mathematics, and computer science.

Computational Methods in Environmental Fluid Mechanics

Computational Fluid Dynamics (CFD) is an important design tool in engineering and also a substantial research tool in various physical sciences as well as in biology. The objective of this book is to provide university students with a solid foundation for understanding the numerical methods employed in today's CFD and to familiarise them with modern CFD codes by hands-on experience. It is also intended for engineers and scientists starting to work in the field of CFD or for those who apply CFD codes. Due to the detailed index, the text can serve as a reference handbook too. Each chapter includes an extensive bibliography, which provides an excellent basis for further studies.

Computational Fluid Dynamics and Heat Transfer

This pioneering text provides an excellent introduction to CFD at the senior level in aerospace and mechanical engineering, and to some extent, chemical and civil engineering. It can also serve as a one-semester introductory course at the beginning graduate level, as a useful precursor to a more serious study of CFD in advanced books. It is presented in a very readable, informal, enjoyable style.

Computational Fluid Dynamics in Food Processing

Fire and combustion presents a significant engineering challenge to mechanical, civil and dedicated fire engineers, as well as specialists in the process and chemical, safety, buildings and structural fields. We are reminded of the tragic outcomes of 'untenable' fire disasters such as at King's Cross underground station or Switzerland's St Gotthard tunnel. In these and many other cases, computational fluid dynamics (CFD) is at the forefront of active research into unravelling the probable causes of fires and helping to design structures and systems to ensure that they are less likely in the future. Computational fluid dynamics (CFD) is routinely used as an analysis tool in fire and combustion engineering as it possesses the ability to handle the complex geometries and characteristics of combustion and fire. This book shows engineering students and professionals how to understand and use this powerful tool in the study of combustion processes, and in the engineering of safer or more fire resistant (or conversely, more fire-efficient) structures. No other book is dedicated to computer-based fire dynamics tools and systems. It is supported by a rigorous pedagogy, including worked examples to illustrate the capabilities of different models, an introduction to the essential aspects of fire physics, examination and self-test exercises, fully worked solutions and a suite of accompanying software for use in industry standard modeling systems. · Computational Fluid Dynamics (CFD) is widely used in engineering analysis; this is the only book dedicated to CFD modeling analysis in fire and combustion engineering · Strong pedagogic features mean this book can be used as a text for graduate level mechanical, civil, structural and fire engineering courses, while its coverage of the latest techniques and industry standard software make it an important reference for researchers and professional engineers in the mechanical and structural sectors, and by fire engineers, safety consultants and regulators · Strong author team (CUHK is a recognized centre of excellence in fire eng) deliver an expert package for students and professionals, showing both

theory and applications. Accompanied by CFD modeling code and ready to use simulations to run in industry-standard ANSYS-CFX and Fluent software.

Computational Fluid Dynamics

Using HPC for Computational Fluid Dynamics

This book offers a practical, application-oriented introduction to computational fluid dynamics (CFD), with a focus on the concepts and principles encountered when using CFD in industry. Presuming no more knowledge than college-level understanding of the core subjects, the book puts together all the necessary topics to give the reader a comprehensive introduction to CFD. It includes discussion of the derivation of equations, grid generation and solution algorithms for compressible, incompressible and hypersonic flows. The final two chapters of the book are intended for the more advanced user. In the penultimate chapter, the special difficulties that arise while solving practical problems are addressed. Distinction is made between complications arising out of geometrical complexity and those arising out of the complexity of the physics (and chemistry) of the problem. The last chapter contains a brief discussion of what can be considered as the Holy Grail of CFD, namely, finding the optimal design of a fluid flow component. A number of problems are given at the end of each chapter to reinforce the concepts and ideas discussed in that chapter. CFD has come of age and is widely used in industry as well as in academia as an analytical tool to investigate a wide range of fluid flow problems. This book is written for two groups: for those students who are encountering CFD for the first time in the form of a taught lecture course, and for those practising engineers and scientists who are already using CFD as an analysis tool in their professions but would like to deepen and broaden their understanding of the subject.

Applied and Computational Fluid Mechanics

This text describes several computational techniques that can be applied to a variety of problems in thermo-fluid physics, multi-phase flow, and applied mechanics involving moving flow boundaries. 1996 edition.

Introduction to Computational Fluid Dynamics

Fluids play an important role in environmental systems appearing as surface water in rivers, lakes, and coastal regions or in the subsurface as well as in the atmosphere. Mechanics of environmental fluids is concerned with fluid motion, associated mass and heat transport as well as deformation processes in subsurface systems. In this reference work the fundamental modelling approaches based on continuum mechanics for fluids in the environment are described, including porous media and turbulence. Numerical methods for solving the process governing equations as well as its object-oriented computer implementation are discussed and illustrated with examples. Finally, the application of computer models in civil and environmental engineering is demonstrated.

Computational Fluid Dynamics in Industrial Combustion

Computational Fluid Dynamics: Principles and Applications

Computational Fluid Dynamics Simulation of Spray Dryers

An introduction to CFD fundamentals and using 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 screen shots, and step by step processes, this book walks the reader through modeling and computing, as well as interpreting CFD results. The first book in the field aimed at CFD users rather than developers. New to this edition: A more comprehensive coverage of CFD techniques including discretisation via finite element and spectral element as well as finite difference and finite volume methods and multigrid method. Coverage of different approaches to CFD grid generation in order to closely match how CFD meshing is being used in industry. Additional coverage of high-pressure fluid dynamics and meshless approach to provide a broader overview of the application areas where CFD can be used. 20% new content

Computational Fluid Dynamics for Engineers

The implementation of early-stage simulation tools, specifically computational fluid dynamics (CFD), is an international and interdisciplinary trend that allows engineers to computer-test concepts all the way through the development of a process or system. With the enhancement of computing power and efficiency, and the availability of affordable CFD packages, the applications of CFD have extended into the food industry for modeling industrial processes, performing comprehensive analyses, and optimizing the efficiency and cost effectiveness of the new processes and systems. Beginning a new series dedicated to contemporary, up-to-date food engineering practices, Computational Fluid Dynamics in Food Processing is the first book of its kind to illustrate the use of CFD for solving heat and mass transfer problems in the food industry. Using a computational grid, CFD solves governing equations that describe fluid flow across each grid cell by means of an iterative procedure in order to predict and visualize the profiles of velocity, temperature, pressure, and other parameters. Starting with an overview of CFD technology and applications, the book illustrates the use of CFD for gaining a qualitative and quantitative assessment of the performance of processes involving heat and mass transfer. Specific chapters cover airflow in refrigerated trucks, retail display cabinets, microwaves, and doorways; velocity in meat dryers and spray drying; thermal sterilization; plate heat exchangers; membrane separation systems; jet impingement ovens; food extrusion and high-pressure processing; prediction of hygiene; design of biosensors; and the fermentation of tea and ripening of cheese. Drawing from an esteemed panel of international professionals and academics, this groundbreaking bookprovides engineers and technologists in research, development, and operations with critical, comprehensive, and readily accessible information on the art and science of CFD technology.

Environmental Modelling

Computational Fluid-Structure Interaction: Methods, Models, and Applications provides detailed explanations of a range of FSI models, their mathematical formulations, validations, and applications, with an emphasis on conservative unstructured-grid FVM. The first part of the book presents the nascent numerical methods, algorithms and solvers for both compressible and incompressible flows, computational structural dynamics (CSD), parallel multigrid, IOM, IMM and ALE methods. The second half covers the validations of these numerical methods and solvers, as well as their applications in a broad range of areas in basic research and engineering. Provides a comprehensive overview of the latest numerical methods used in FSI, including the unstructured-grid finite volume method (FVM), parallel multigrid scheme, overlapping mesh, immersed object method (IOM), immersed membrane method (IMM), arbitrary Lagragian-Eulerian (ALE), and more Provides full details of the numerical methods, solvers and their validations Compares different methods to help readers more effectively choose the right approach for their own FSI problems Features real-life FSI case studies, such as large eddy simulation of aeroelastic flutter of a wing, parallel computation of a bio-prosthetic heart valve, and ALE study of a micro aerial vehicle

Advances of Computational Fluid Dynamics in Nuclear Reactor Design and Safety Assessment

This book provides an introduction, overview, and specific examples of computational fluid dynamics and their applications in the water, wastewater, and stormwater industry.

Computational Fluid Dynamics

This textbook explores both the theoretical foundation of the Finite Volume Method (FVM) and its applications in Computational Fluid Dynamics (CFD). Readers will discover a thorough explanation of the FVM numerics and algorithms used for the simulation of incompressible and compressible fluid flows, along with a detailed examination of the components needed for the development of a collocated unstructured pressure-based CFD solver. Two particular CFD codes are explored. The first is uFVM, a three-dimensional unstructured pressure-based finite volume academic CFD code, implemented within Matlab. The second is OpenFOAM®, an open source framework used in the development of a range of CFD programs for the simulation of industrial scale flow problems. With over 220 figures, numerous examples and more than one hundred exercise on FVM numerics, programming, and applications, this textbook is suitable for use in an introductory course on the FVM, in an advanced course on numerics, and as a reference for CFD programmers and researchers.

Applied Computational Fluid Dynamics and Turbulence Modeling

Covered from the vantage point of a user of a commercial flow package, Essentials of Computational Fluid Dynamics provides the information needed to competently operate a commercial flow solver. This book provides a physical description of fluid flow, outlines the strengths and weaknesses of computational fluid dynamics

(CFD), presents the basics of the discretization of the equations, focuses on the understanding of how the flow physics interact with a typical finite-volume discretization, and highlights the approximate nature of CFD. It emphasizes how the physical concepts (mass conservation or momentum balance) are reflected in the CFD solutions while minimizing the required mathematical/numerical background. In addition, it uses cases studies in mechanical/aero and biomedical engineering, includes MATLAB and spreadsheet examples, codes and exercise questions. The book also provides practical demonstrations on core principles and key behaviors and incorporates a wide range of colorful examples of CFD simulations in various fields of engineering. In addition, this author: Introduces basic discretizations, the linear advection equation, and forward, backward and central differences Proposes a prototype discretization (first-order upwind) implemented in a spreadsheet/MATLAB example that highlights the diffusive character Looks at consistency, truncation error, and order of accuracy Analyzes the truncation error of the forward, backward, central differences using simple Taylor analysis Demonstrates how the of upwinding produces Artificial Viscosity (AV) and its importance for stability Explains how to select boundary conditions based on physical considerations Illustrates these concepts in a number of carefully discussed case studies Essentials of Computational Fluid Dynamics provides a solid introduction to the basic principles of practical CFD and serves as a resource for students in mechanical or aerospace engineering taking a first CFD course as well as practicing professionals needing a brief, accessible introduction to CFD.

Computational Fluid-Structure Interaction

Computational Fluid-Structure Interaction: Methods andApplications takes the reader from the fundamentals ofcomputational fluid and solid mechanics to the state-of-the-art incomputational FSI methods, special FSI techniques, and solution ofreal-world problems. Leading experts in the field present thematerial using a unique approach that combines advanced methods,special techniques, and challenging applications. This book begins with the differential equations governing thefluid and solid mechanics, coupling conditions at thefluid–solid interface, and the basics of the finite elementmethod. It continues with the ALE and space–time FSI methods,spatial discretization and time integration strategies for thecoupled FSI equations, solution techniques for thefully-discretized coupled equations, and advanced FSI andspace–time methods. It ends with special FSI techniquestargeting cardiovascular FSI, parachute FSI, and wind-turbineaerodynamics and FSI. Key features: First book to address the state-of-the-art in computationalFSI Combines the fundamentals of computational fluid and solidmechanics, the state-of-the-art in FSI methods, and specialFSI techniques targeting challenging classes of real-worldproblems Covers modern computational mechanics techniques, includingstabilized, variational multiscale, and space–time methods,isogeometric analysis, and advanced FSI coupling methods Is in full color, with diagrams illustrating the fundamentalconcepts and advanced methods and with insightful visualizationillustrating the complexities of the problems that can be solvedwith the FSI methods covered in the book. Authors are award winning, leading global experts incomputational FSI, who are known for solving some of the mostchallenging FSI problems Computational Fluid-Structure Interaction: Methods andApplications is a comprehensive reference for researchers andpracticing

engineers who would like to advance their existingknowledge on these subjects. It is also an ideal text for graduateand senior-level undergraduate courses in computational fluidmechanics and computational FSI.

Computational Fluid-Structure Interaction

Computational fluid dynamics (CFD) combines continuum and discrete theories for fluid modeling with computational algorithms for fluid simulation. It is an important research area since there is a wide range of natural phenomena that can be modeled through fluid theory. Some common engineering examples are pumps, fans, turbines, airplanes, ships, rivers, windmills, pipes, and more recently, the hemodynamics of the arterial system. This book examines several characteristics of CFD, as well as its applications and analysis.

Computational Fluid Dynamics

Although many books have been written on computational fluid dynamics (CFD) and many written on combustion, most contain very limited coverage of the combination of CFD and industrial combustion. Furthermore, most of these books are written at an advanced academic level, emphasize theory over practice, and provide little help to engineers who need to use CFD for combustion modeling. Computational Fluid Dynamics in Industrial Combustion fills this gap in the literature. Focusing on topics of interest to the practicing engineer, it codifies the many relevant books, papers, and reports written on this combined subject into a single, coherent reference. It looks at each topic from a somewhat narrow perspective to see how that topic affects modeling in industrial combustion. The editor and his team of expert authors address these topics within three main sections: Modeling Techniques-The basics of CFD modeling in combustion Industrial Applications-Specific applications of CFD in the steel, aluminum, glass, gas turbine, and petrochemical industries Advanced Techniques-Subjects rarely addressed in other texts, including design optimization, simulation, and visualization Rapid increases in computing power and significant advances in commercial CFD codes have led to a tremendous increase in the application of CFD to industrial combustion. Thorough and clearly representing the techniques and issues confronted in industry, Computational Fluid Dynamics in Industrial Combustion will help bring you quickly up to date on current methods and gain the ability to set up and solve the various types of problems you will encounter.

Computational Fluid Dynamics in Fire Engineering

This book is primarily for a first one-semester course on CFD; in mechanical, chemical, and aeronautical engineering. Almost all the existing books on CFD assume knowledge of mathematics in general and differential calculus as well as numerical methods in particular; thus, limiting the readership mostly to the postgraduate curriculum. In this book, an attempt is made to simplify the subject even for readers who have little or no experience in CFD, and without prior knowledge of fluid-dynamics, heattransfer and numerical-methods. The major emphasis is on simplification of the mathematics involved by presenting physical-law (instead of the traditional differential equations) based algebraic-formulations,

discussions, and solution-methodology. The physical law based simplified CFD approach (proposed in this book for the first time) keeps the level of mathematics to school education, and also allows the reader to intuitively get started with the computer-programming. Another distinguishing feature of the present book is to effectively link the theory with the computer-program (code). This is done with more pictorial as well as detailed explanation of the numerical methodology. Furthermore, the present book is structured for a module-by-module code-development of the two-dimensional numerical formulation; the codes are given for 2D heat conduction, advection and convection. The present subject involves learning to develop and effectively use a product - a CFD software. The details for the CFD development presented here is the main part of a CFD software. Furthermore, CFD application and analysis are presented by carefully designed example as well as exercise problems; not only limited to fluid dynamics but also includes heat transfer. The reader is trained for a job as CFD developer as well as CFD application engineer; and can also lead to start-ups on the development of "apps" (customized CFD software) for various engineering applications. "Atul has championed the finite volume method which is now the industry standard. He knows the conventional method of discretizing differential equations but has never been satisfied with it. As a result, he has developed a principle that physical laws that characterize the differential equations should be reflected at every stage of discretization and every stage of approximation. This new CFD book is comprehensive and has a stamp of originality of the author. It will bring students closer to the subject and enable them to contribute to it." —Dr. K. Muralidhar, IIT Kanpur, INDIA

ROMANCE ACTION & ADVENTURE MYSTERY & THRILLER BIOGRAPHIES & HISTORY CHILDREN'S YOUNG ADULT FANTASY HISTORICAL FICTION HORROR LITERARY FICTION NON-FICTION SCIENCE FICTION