Cfd Modelling Of Hydrodynamics And Heat Transfer In

Multiscale Modeling for Process Safety Applications is a new reference demonstrating the implementation of multiscale modeling techniques on process safety applications. It is a valuable resource for readers interested in theoretical simulations and/or computer simulations of hazardous scenarios. As multi-scale modeling is a computational technique for solving problems involving multiple scales, such as how a flammable vapor cloud might behave if ignited, this book provides information on the fundamental topics of toxic, fire, and air explosion modeling, as well as modeling jet and pool fires using computational fluid dynamics. The book goes on to cover nanomaterial toxicity, QPSR analysis on relation of chemical structure to flash point, molecular structure and burning velocity, first principle studies of reactive chemicals, water and air reactive chemicals, and dust explosions. Chemical and process safety professionals, as well as faculty and graduate researchers, will benefit from the detailed coverage provided in this book. Provides the only comprehensive source addressing the use of multiscale modeling in the context of process safety Bridges multiscale modeling with process safety, enabling the reader to understand mapping between problem detail and effective usage of resources

Presents an overall picture of addressing safety problems in all levels of modeling and the latest approaches to each in the field Features worked out examples, case studies, and a question bank to aid understanding and involvement for the reader
Mixing reactor is a common unit operation in many industrial processes. It is often encountered multiphase flow for mixing process. A computational fluid dynamics (CFD) model based on the mixture formulation for multiphase flow was created to study the mixing hydrodynamics of oil and water in a mixing reactor. This project aimed to develop a reliable CFD model that was able to predict the hydrodynamic properties like velocity or volume fraction. A grid dependency study was conducted to determine the effect of coarse, medium and fine meshes. The simulations were based on an experimental study where the mixing of oil and water occurred in the external loop. It had shown that the medium mesh was seem satisfactory to fulfill the evaluation criterion. The error between prediction of CFD model and experimental data was also within the tolerant range.
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 Since many processes in the food industry involve fluid flow and heat and mass transfer, Computational Fluid Dynamics (CFD) provides a powerful early-stage simulation tool for gaining a qualitative and quantitative assessment of the performance of food processing, allowing engineers to test concepts all the way through the development of a process or system. Published in 2007, the first edition was the first book to address the use of CFD in food processing applications, and its aims were to present a comprehensive review of CFD applications for the food industry and pinpoint the research and development trends in the development of the technology; to provide the engineer and technologist working in research, development, and operations in the food industry with critical, comprehensive, and readily accessible information on the art and science of CFD; and to serve as an essential reference source to undergraduate and postgraduate students and researchers in universities and research institutions. This will continue to be the purpose of this second edition. In the second edition, in order to reflect the most recent research and development trends in the technology, only a few original chapters are updated with the latest developments. Therefore, this new edition mostly contains new chapters covering the analysis and optimization of cold chain facilities, simulation of thermal processing and modeling of heat exchangers, and CFD applications in other food processes.

This book presents new food production systems (for plants and animals) involving agrochemicals that increase in a controlled manner the bioactives content, under greenhouse conditions. Moreover, conception and design of new instrumentation for precision agriculture and aquiculture contributing in food production is also highlighted in this book.

This volume contains the lectures presented at the

NATO Advanced Study Institute that took place at the University of Delaware, Newark, Delaware, July 18-27, 1982. The purpose of this Institute was to provide an international forum for exchange of ideas and dissemination of knowledge on some selected topics in Mechanics of Fluids in Porous Media. Processes of transport of such extensive quantities as mass of a phase, mass of a component of a phase, momentum and/or heat occur in diversified fields, such as petroleum reservoir engineer ing, groundwater hydraulics, soil mechanics, industrial filtration, water purification, wastewater treatment, soil drainage and irri gation, and geothermal energy production. In all these areas, scientists, engineers and planners make use of mathematical models that describe the relevant transport processes that occur within porous medium domains, and enable the forecasting of the future state of the latter in response to planned activities. The mathe matical models, in turn, are based on the understanding of phenomena, often within the void space, and on theories that re late these phenomena to measurable quantities. Because of the pressing needs in areas of practical interest, such as the develop ment of groundwater resources, the control and abatement of groundwater contamination, underground energy storage and geo thermal energy production, a vast amount of research efforts in all these fields has contributed, especially in the last t~o decades, to our understanding and ability to describe transport phenomena.

"Vive la Revolution!" was the theme of the Twenty-Third Symposium on Naval Hydrodynamics held in Val de Reuil, France, from September 17-22, 2000 as more than 140 experts in ship design, construction, and operation came together to exchange naval research developments. The forum encouraged both formal and informal discussion of presented papers, and the occasion provides an opportunity for direct communication between international peers. This book includes sixty-three papers presented at the symposium which was organized jointly by the Office of Naval Research, the National Research Council (Naval Studies Board), and the Bassin d'Essais des Carènes. This book includes the ten topical areas discussed at the symposium: wave-induced motions and loads, hydrodynamics in ship design, propulsor hydrodynamics and hydroacoustics, CFD validation, viscous ship hydrodynamics, cavitation and bubbly flow, wave hydrodynamics, wake dynamics, shallow water hydrodynamics, and fluid dynamics in the naval context.

This book gives an overview of recent integrated and inter-disciplinary approaches between chemical experiment and theory in a variety of fields, from polymer science to materials chemistry and ranging from the design of tailored properties to catalysis and reactivity, building on the well-established success of Density Functional Theory as the foremost quantum chemical method to provide qualitative and quantitative interpretation of results from the chemical laboratory. The combination of several characterization techniques with an understanding at the molecular level of chemical and physical phenomena are the main focal point of the subject matter.

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

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.

Dams and their auxiliary structures are built to provide water for human consumption, irrigating lands, generating hydroelectric power, and use in industrial processes. They are critical structures for continuing life and providing public safety. Construction of a dam is a complicated task that requires sophisticated modern technology and technical expertise. Scientists need to review and adjust their perspectives on designing embankments and their related structures, and compaction and consolidation of fill material, behavior of concrete materials, geotechnical and seismological studies of the dam site, total risk analysis, safety monitoring and instrumentation, heightening, hydrological studies, soil conservation, and watershed management. This book intends to provide the reader with a comprehensive overview of the latest information in dam engineering.

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.

Bubble columns are widely used as gas- iquid contactors and as reactors in chemical, petrochemical and biochemical industries. Effective mixing as well as high interfacial area between the phases, leading to improved heat and mass transfer characteristics, relatively cheap to install and the lack of moving parts, are the factors that render under bubble columns an attractive choice as reactors for the described processes. Gas-liquid flow in bubble column reactors is characterized by a combination of inherently unsteady complex processes with widely varying spatial and temporal scales. Understanding the complexity of the fluid dynamics and mass transfer in bubble column and is important due to its application in the chemical and bioprocess industries. The potential of Computational Fluid Dynamics (CFD) for describing the hydrodynamics and heat and mass transfer of bubble columns has been established by several publications in the past. CFD predicts what happens quantitatively, when fluids flow, often with the complications of simultaneous flow of heat, mass transfer (eg perspiration, dissolution), phase change (eg melting, freezing, boiling), chemical reaction (eg combustion, rusting), mechanical movement (eg of pistons, fans, rudders), stresses in and displacement of immersed or surrounding solids. Thus CFD can successfully be used to study the gas-liquid mass transfer in bubble column reactor. In the present work an attempt has been made to understand the hydrodynamic behavior and gas-liquid mass transfer (transfer of oxygen from air to de-aerated water) of a concurrent gas(air)-liquid(water) up-flow bubble column by CFD analysis. The system used in the study is a cylindrical column of 10 cm ID and 1.88 m height. GAMBIT 2.3.16 has been used to generate a 2D coarse grid of 0.01m by 0.01m mesh size. The eulerian-eulerian approach has been used for modeling the multiphase flow and the oxygen mass transfer from air to de-aerated water and the column hydrodynamics. The.

This book covers a wide variety of topics related to advancements in different stages of mass transfer modelling processes. Its purpose is to create a platform for the exchange of recent observations, experiences, and achievements. It is recommended for those in the chemical, biotechnological, pharmaceutical, and nanotechnology industries as well as for students of natural sciences, technical, environmental and employees in companies which manufacture machines for the above-mentioned industries. This work can also be a useful source for researchers and engineers dealing with mass transfer and related issues.

All over the world sport plays a prominent role in society: as a leisure activity for many, as an ingredient of culture, as a business and as a matter of national prestige in such major events as the World Cup in soccer or the Olympic Games. Hence, it is not surprising that science has entered the realm of sports, and, in particular, that computer simulation has become highly relevant in recent years. This is explored in this book by choosing five different sports as examples, demonstrating that computational science and engineering (CSE) can make essential contributions to research on sports topics on both the fundamental level and, eventually, by supporting athletes' performance.

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.

In this study we develop the computational and experimental tools to assist us in performance evaluation of trickle bed reactors (TBRs). The study focuses on experimental characterization of the flow distribution, and development of computational fluid dynamics (CFD) model of trickle flow. The experimental study has been performed to examine the quality of liquid phase distribution in a high pressure system. The results were provided in terms of distribution of the effluent liquid fluxes and cross-sectional liquid holdups. Their individual trends, but also their relation with respect to operating conditions was examined. Characterization of bed porosity distribution has been performed and used as the input to the computational model. The experimental study of the dependence of the extent of hysteresis on operating parameters in a high pressure TBR was performed. The extent of hysteresis was found uniquely determined by the pressure drop in the Levec prewetting mode. This fact and developed CFD model were then used to deduce conditions leading to operation with negligible hysteresis effects. Three-dimensional Eulerian CFD model is developed. Phase interaction closures are based on the film flow model, principles of statistical hydrodynamics and relative permeability concept. Model has been assessed against experimental data for liquid holdup, wetting efficiency and pressure drop hysteresis. Hydrodynamic Eulerian CFD model is then used together with species balance to examine the TBR performance for gas and liquid reactant limited systems. For each case a closed form approach of coupling bed and particle scale solution within CFD framework was presented.

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.

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 (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. ? 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.

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.

The objective of this project is to simulate a gas-solid fluidized by applying CFD techniques in order to investigate hydrodynamics and heat transfer phenomena. Reactor model predictions will be compared with the corresponding experimental data reported in the literature to validate the model . To simulate a gas-solid fluidized bed we need to use the multiphase flow approach . First we have to write the equations for the different flow regimes and then different CFD techniques are applied for discretization of those equations. After that a code is written for calculating the values of volume fraction, velocity and temperature.

The aim of this work is to comparing the simulation result from Computational Fluid Dynamic (CFD) with the experimental measurement from Montante et al. (2007). Liquid axial velocity and liquid radial velocity were measured in order to get the similar result with experimental measurement from Montante et al. (2007). For this work, Gambit 2.4.6 and Fluent 6.3.26 software were use in modelling and running the simulation. The experimental results obtained at different gas flow rates are presented, compared with multi-phase data and discussed for gaining insight into the gas-liquid flows. The agreement between the experimental and the calculated mean velocity fields indicates that the selected CFD modelling is appropriate for the prediction of the mean hydrodynamic features of gas-liquid dispersions in stirred vessels.

With the advancement of computers, the use of modeling to reduce time and expense, and improve process optimization, predictive capability, process automation, and control possibilities, is now an integral part of food science and engineering. New technology and ease of use expands the range of techniques that scientists and researchers have at the

Over the past few decades there has been a prolific increase in research and development in area of heat transfer, heat exchangers and their associated technologies. This book is a collection of current research in the above mentioned areas and describes modelling, numerical methods, simulation and information technology with modern ideas and methods to analyse and enhance heat transfer for single and multiphase systems. The topics considered include various basic concepts of heat transfer, the fundamental modes of heat transfer (namely conduction, convection and radiation), thermophysical properties, computational methodologies, control, stabilization and optimization problems, condensation, boiling and freezing, with many real-world problems and important modern applications. The book is divided in four sections : "Inverse, Stabilization and Optimization Problems", "Numerical Methods and Calculations", "Heat Transfer in Mini/Micro Systems", "Energy Transfer and Solid Materials", and each section discusses various issues, methods and applications in accordance with the subjects. The combination of fundamental approach with many important practical applications of current interest will make this book of interest to researchers, scientists, engineers and graduate students in many disciplines, who make use of mathematical modelling, inverse problems, implementation of recently developed numerical methods in this multidisciplinary field as well as to experimental and theoretical researchers in the field of heat and mass transfer.

This Element presents a unified computational fluid dynamics framework from rarefied to continuum regimes. The framework is based on the direct modelling of flow physics in a discretized space. The mesh size and time step are used as modelling scales in the construction of discretized governing equations. With the variation-of-cell Knudsen number, continuous modelling equations in different regimes have been obtained, and the Boltzmann and Navier-Stokes equations become two limiting equations in the kinetic and hydrodynamic scales. The unified algorithms include the discrete velocity method (DVM)–based unified gas-kinetic scheme (UGKS), the particlebased unified gas-kinetic particle method (UGKP), and the wave and particle–based unified gas-kinetic wave-particle method (UGKWP). The UGKWP is a multi-scale method with the particle for non-equilibrium transport and wave for equilibrium evolution. The particle dynamics in the rarefied regime and the hydrodynamic flow solver in the continuum regime have been unified according to the cell's Knudsen number.

The applications of bubble columns are very important as multiphase contactors and reactors in process industry. They are wide and extensively used in chemical, petrochemical and biochemical industries. The advantages of bubble column are low maintenance and operating cost due to the compactness and no moving part. They also have an excellent mass and heat transfer characteristic or high heat and mass transfer coefficients, and high durability of catalyst or packing material. It is important to understand the nature of hydrodynamics and operational parameters to characterize their operation including pressure drop, gas superficial velocity, bubble rise velocity, etc., to do the design and scale-up process. Although experimental methods are available to elucidate the multiphase flow in bubble column by the means of advanced experimental methods i.e. X-ray tomography and laser doppler anemometry, the experimental setup is often expensive to develop. Alternatively, the computational fluid dynamics can be used to evaluate the performance of bubble column at lower cost compared to experimental setup. In this work commercial CFD software, FLUENT 6.3 was employed for modeling of gasliquid flow in a bubble column. Multiphase simulations were performed using an Eulerian-Eulerian two-fluid model and the drag coefficient of spherical and distorted bubbles was modeled using the Tomiyama (1995) and Schiller-Naumann (1935) models. The effect of the void fractions on the drag coefficient was modeled using the correlation by Behzadi (2004). The CFD predictions were compared to the experimental measurement adopted from literature. The CFD predicts the turbulent kinetic energy, gas hold-up and the liquid axial velocity fairly well, although the results seem to suggest that further improvement on the interfacial exchange models and possibly further refinement on the two-fluid modeling approaches are necessary especially for the liquid axial velocity and turbulent kinetic energy. It is clear from the modeling exercise performed in this work that CFD is a great method for modeling the performance of bubble column. Furthermore, the CFD method is certainly less expensive than the experimental characterization studies.

This study aims to present the results obtained from the simulation of a flat plate photocatalytic reactor using CFD code FLUENT. For various turbulence models, the simulation results showed the computed flow features for inlet and outlet components of the flat plate reactor under different flow regimes.In addition the performance of the photocatalytic reactor for pollutant degradation was observed to depend on the reactor's hydrodynamics. The results reported here suggest the importance of fluid mixing in the flat plate reactor since the reaction takes place only at the fluid-catalyst interface.The effect of inlet positions and roughness elements on the flow and mass transport of formic acid in the reactive module has been examined which provides significant insight for the efficient design of the flat plate reactor. This more-of-physics, less-of-math, insightful and comprehensive book simplifies computational fluid dynamics for readers with little knowledge or experience in heat transfer, fluid dynamics or numerical methods. The novelty of this book lies in the simplification of the level of mathematics in CFD by presenting physical law (instead of the traditional differential equations) and discrete (independent of continuous) math-based algebraic formulations. Another distinguishing feature of this book is that it effectively links theory with computer program (code). This is done with pictorial as well as detailed explanations of implementation of the numerical methodology. It also includes pedagogical aspects such as end-of-chapter problems and carefully designed examples to augment learning in CFD code-development, application and analysis. This book is a valuable resource for students in the fields of mechanical, chemical or aeronautical engineering. In this Special Issue, one review paper highlights the necessity of multiscale CFD, coupling micro- and macro-scales, for exchanging information at the interface of the two scales. Four research papers investigate the hydrodynamics, heat transfer, and chemical reactions of various processes using Eulerian CFD modeling. CFD models are attractive for industrial applications. However, substantial efforts in physical modeling and numerical implementation are still required before their widespread implementation.

This book is served as a reference text to meet the needs of advanced scientists and research engineers who seek for their own computational fluid dynamics (CFD) skills to solve a variety of fluid flow problems. Key Features: - Flow Modeling in Sedimentation Tank, - Greenhouse Environment, - Hypersonic Aerodynamics, - Cooling Systems Design, - Photochemical Reaction Engineering, - Atmospheric Reentry Problem, - Fluid-Structure Interaction (FSI), - Atomization, - Hydraulic Component Design, - Air Conditioning System, - Industrial Applications of CFD

Instabilities of fluid flows and the associated transitions between different possible flow states provide a fascinating set of problems that have attracted researchers for over a hundred years. This book addresses state-of-the-art developments in numerical techniques for computational modelling of fluid instabilities and related bifurcation structures, as well as providing comprehensive reviews of recently solved challenging problems in the field.

Computational fluid dynamics (CFD), which uses numerical analysis to predict and model complex flow behaviors and transport processes, has become a mainstream tool in engineering process research and development. Complex chemical processes often involve coupling between dynamics at vastly different length and time scales, as well as coupling of different physical models. The multiscale and multiphysics nature of those problems calls for delicate modeling approaches. This book showcases recent contributions in this field, from the development of modeling methodology to its application in supporting the design, development, and optimization of engineering 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

In this translation of the German edition, the authors provide insight into the numerical simulation of fluid flow. Using a simple numerical method as an expository example, the individual steps of scientific computing are presented: the derivation of the mathematical model; the discretization of the model equations; the development of algorithms; parallelization; and visualization of the computed data. In addition to the treatment of the basic equations for modeling laminar, transient flow of viscous, incompressible fluids - the Navier-Stokes equations - the authors look at the simulation of free surface flows; energy and chemical transport; and turbulence. Readers are enabled to write their own flow simulation program from scratch. The variety of applications is shown in several simulation results, including 92 black-and-white and 18 color illustrations. After reading this book, readers should be able to understand more enhanced algorithms of computational fluid dynamics and apply their new knowledge to other scientific fields.

Computational Fluid DynamicsA Practical ApproachButterworth-Heinemann