Cfd Supported Modelling Of Double Skin Facades In Hot Arid

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.

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.

Explores and brings together the existent body of knowledge on building performance analysis Building performance is an important yet surprisingly complex concept. This book presents a comprehensive and systematic overview of the subject. It provides a working definition of building performance, and an in-depth discussion of the role building performance plays throughout the building life cycle. The book also explores the perspectives of various stakeholders, the functions of buildings, performance requirements, performance quantification (both predicted and measured), criteria for success, and the challenges of using performance analysis in practice. Building Performance Analysis starts by introducing the subject of building performance: its key terms, definitions, history, and challenges. It then develops a theoretical foundation for the subject, explores the complexity of performance assessment, and the way that performance analysis impacts on actual buildings. In doing so, it attempts to answer the following questions: What is building performance? How can building performance be measured and analyzed? How does the analysis of building

performance guide the improvement of buildings? And what can the building domain learn from the way performance is handled in other disciplines? Assembles the current body of knowledge on building performance analysis in one unique resource Offers deep insights into the complexity of using building performance analysis throughout the entire building life cycle, including design, operation and management Contributes an emergent theory of building performance and its analysis Building Performance Analysis will appeal to the building science community, both from industry and academia. It specifically targets advanced students in architectural engineering, building services design, building performance simulation and similar fields who hold an interest in ensuring that buildings meet the needs of their stakeholders.

The ongoing digitalization of the energy sector, which will make a large amount of data available, should not be viewed as a passive ICT application for energy technology or a threat to thermodynamics and fluid dynamics, in the light of the competition triggered by data mining and machine learning techniques. These new technologies must be posed on solid bases for the representation of energy systems and fluid machinery. Therefore, mathematical modelling is still relevant and its importance cannot be underestimated. The aim of this Special Issue was to collect contributions about

mathematical modelling of energy systems and fluid machinery in order to build and consolidate the base of this knowledge.

In addition to the application of fundamental principles that lead to a structured method for zero carbon design of buildings, this considerably expanded second edition includes new advanced topics on multi-objective optimisation; reverse modelling; reduction of the simulation performance gap; predictive control; nature-inspired emergent simulation leading to sketches that become 'alive'; and an alternative economics for achieving the sustainability paradigm. The book features student design work from a Master's programme run by the author, and their design speculation for a human settlement on Mars. Tasks for simple simulation experiments are available for the majority of topics, providing the material for classroom exercise and giving the reader an easy introduction into the field. Extended new case studies of zero carbon buildings are featured in the book, including schemes from Japan, China, Germany, Denmark and the UK, and provide the reader with an enhanced design toolbox to stimulate their own design thinking.

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.

Issues in Extreme Conditions Technology Research and Application: 2011 Edition is a ScholarlyEditions™ eBook that delivers timely, authoritative, and comprehensive information about Extreme Conditions Technology Research and Application. The editors have built Issues in Extreme Conditions Technology Research and Application: 2011 Edition on the vast information databases of ScholarlyNews.™ You can expect the information about Extreme Conditions Technology Research and Application in this eBook to be deeper than what you can access anywhere else, as well as consistently reliable, authoritative, informed, and relevant. The content of Issues in Extreme Conditions Technology Research and Application: 2011 Edition has been produced by the world's leading scientists, engineers, analysts, research institutions, and companies. All of the content is from peer-reviewed sources, and all of it is written, assembled, and edited by the editors at ScholarlyEditions™ and available exclusively from us. You now have a source you can cite with authority, confidence, and credibility. More information is available at http://www.ScholarlyEditions.com/.

Prof. D. Brian Spalding, working with a small group of students and colleagues at Imperial College, London in the mid-to late-1960's, single-handedly pioneered the use of Computational Fluid Dynamics (CFD) for engineering practice.?This book brings together advances in computational fluid dynamics in a collection of chapters

authored by leading researchers, many of them students or associates of Prof. Spalding. The book intends to capture the key developments in specific fields of activity that have been transformed by application of CFD in the last 50 years. The focus is on review of the impact of CFD on these selected fields and of the novel applications that CFD has made possible. Some of the chapters trace the history of developments in a specific field and the role played by Spalding and his contributions. The volume also includes a biographical summary of Brian Spalding as a person and as a scientist, as well as tributes to Brian Spalding by those whose life was impacted by his innovations. This volume would be of special interest to researchers, practicing engineers, and graduate students in various fields, including aerospace, energy, power and propulsion, transportation, combustion, management of the environment, health and pharmaceutical sciences.

"Electrostatic Precipitation" includes selected papers presented at the 11th International Conference on Electrostatic Precipitation. It presents the newest developments in electrostatic precipitation, flue gas desulphurization (FGD), selective catalytic reduction (SCR), and non-thermal plasma techniques for multi-pollutants emission control. Almost all outstanding scientists and engineers world-wide in the field will report their on-going researches. The book will be a useful reference for scientists and engineers to keep abreast of the latest developments in environmental science and engineering.

These proceedings collect the papers presented at the 30th International Symposium on Shock Waves (ISSW30), which was held in Tel-Aviv Israel from July 19 to July 24, 2015. The Symposium was organized by Ortra Ltd. The ISSW30 focused on the state of knowledge of the following areas: Nozzle Flow, Supersonic and Hypersonic Flows with Shocks,

Supersonic Jets, Chemical Kinetics, Chemical Reacting Flows, Detonation, Combustion, Ignition, Shock Wave Reflection and Interaction, Shock Wave Interaction with Obstacles, Shock Wave Interaction with Porous Media, Shock Wave Interaction with Granular Media, Shock Wave Interaction with Dusty Media, Plasma, Magnetohyrdrodynamics, Re-entry to Earth Atmosphere, Shock Waves in Rarefied Gases, Shock Waves in Condensed Matter (Solids and Liquids), Shock Waves in Dense Gases, Shock Wave Focusing, Richtmyer-Meshkov Instability, Shock Boundary Layer Interaction, Multiphase Flow, Blast Waves, Facilities, Flow Visualization, and Numerical Methods. The two volumes serve as a reference for the participants of the ISSW30 and anyone interested in these fields.

" The main objective of the COST Action C13 was to increase the knowledge of properties and possibilities of glazing in order to increase the performance of building envelopes, to reduce the energy consumption and to improve the quality of life with respect to interior space, impact on the environment and human welfare. This collection of papers, presented at meetings and workshops of the COST C13 working groups 1 (Architectural Aspects and Design Integration), 2 (Quality of Interior Space) and 3 (Structural Aspects of Glass) are the result of five years of exchange of ideas, experiences and know-how between members, delegates and experts. It represents the body of knowledge from a restricted but representative group of professionals in Europe on the subject of glass building envelopes. The Steel Structures Laboratory at Ecole Polytechnique Fédérale de Lausanne and the research group Façades & Systems of the Faculty of Architecture at Delft University of Technology have taken the initiative to publish these COST C13 papers in order to disseminate the knowledge to the world of glass façade

professionals and to contribute to the development of a new generation of high-performance glass building envelopes. " The proceedings of the international conference "SMSEC2014", a joint conference of the first "Social Modeling and Simulations" and the 10th "Econophysics Colloquium", held in Kobe in November 2014 with 174 participants, are gathered herein. Cutting edge scientific researches on various social phenomena are reviewed. New methods for analysis of big data such as financial markets, automobile traffics, epidemic spreading, world-trades and social media communications are provided to clarify complex interaction and distributions underlying in these social phenomena. Robustness and fragility of social systems are discussed based on agent models and complex network models. Techniques about high performance computers are introduced for simulation of complicated social phenomena. Readers will feel the researchers minds that deep and quantitative understanding will make it possible to realize comprehensive simulations of our whole society in the near future, which will contribute to wide fields of industry also to scientific policy decision.

Introduction to Turbulence ModelingRANS Modelling in ANSYS CFX ; Near Wall Treatment and Heat Transfer ; Industrial Transition Model Formulation in CFX ; Unsteady CFD and Scale-adaptive Simulation (SAS) ; Double Averaged Turbulence Modelling in Eulerian Multi-phase Flows ; Higher Order Models in CFXComputational Fluid Dynamics in Food ProcessingCRC Press

Today's design professionals are faced with challenges on all fronts. They need not only to keep in step with rapid technological changes and the current revolution in design and construction processes, but to lead the industry. This means actively seeking to innovate through design research, raising the bar in building performance and adopting

advanced technologies in their practice. In a constant drive to improve design processes and services, how is it possible to implement innovations? And, moreover, to assimilate them in such a way that design, methods and technologies remain fully integrated? Focusing on innovations in architecture, this book covers new materials and design methods, advances in computational design practices, innovations in building technologies and construction techniques, and the integration of research with design. Moreover, it discusses strategies for integrating innovation into design practices, risks and economic impacts. Through numerous case studies, it illustrates how innovations have been implemented on actual architectural projects, and how design and technical innovations are used to improve building performance, as well as design practices in cutting-edge architectural and engineering firms. Projects of all scales and building types are discussed in the book, ranging from small-scale installations, academic and commercial buildings to large-scale mixed-use, healthcare, civic, academic, scientific research and sports facilities. Work from design firms around the globe and of various scales is discussed in the book, including for example Asymptote Architecture, cepezed, CO Architects, Consarc Architects, FAAB Architektura, Gerber Architekten, HOK, IDOM-ACXT, MAD Architects, Morphosis Architects, SDA | Synthesis Design + Architecture, Studiotrope, Perkins+Will, Richter Dahl Rocha & Associés, Snøhetta, Rob Ley Studio, Trahan Architects, UNStudio and Zaha Hadid Architects, among many others.

As researchers deal with processes and phenomena that are geometrically complex and phenomenologically coupled the demand for high-performance computational fluid dynamics (CFD) increases continuously. The intrinsic nature of

coupled irreversibility requires computational tools that can provide physically meaningful results within a reasonable time. This book collects the state-of-the-art CFD research activities and future R

Effective building performance simulation can reduce the environmental impact of the built environment, improve indoor quality and productivity, and facilitate future innovation and technological progress in construction. It draws on many disciplines, including physics, mathematics, material science, biophysics and human behavioural, environmental and computational sciences. The discipline itself is continuously evolving and maturing, and improvements in model robustness and fidelity are constantly being made. This has sparked a new agenda focusing on the effectiveness of simulation in building life-cycle processes. Building Performance Simulation for Design and Operation begins with an introduction to the concepts of performance indicators and targets, followed by a discussion on the role of building simulation in performance-based building design and operation. This sets the ground for in-depth discussion of performance prediction for energy demand, indoor environmental quality (including thermal, visual, indoor air quality and moisture phenomena), HVAC and renewable system performance, urban level modelling, building operational optimization and automation. Produced in cooperation with the International Building

Performance Simulation Association (IBPSA), and featuring contributions from fourteen internationally recognised experts in this field, this book provides a unique and comprehensive overview of building performance simulation for the complete building life-cycle from conception to demolition. It is primarily intended for advanced students in building services engineering, and in architectural, environmental or mechanical engineering; and will be useful for building and systems designers and operators.

In this project, we have modeled and simulated a Double Skin Facade (hereafter DSF), 6 meters wide and 4 high. An average day was calculated for every month with official weather data and ANSYS FLUENT v15's Solar Calculator. The focus of the project is to help alleviate one of the problems of DSF: heat accumulating inside of the building because radiation transforms into heat, which is then stuck inside due to the glass' thermal isolation properties, similar to a greenhouse. The design consists of a series of Venetian blinds connected by pipes through which water is circulated, accumulating sun's radiated heat and remove it from the façade. In the results section, the amount of heat is analyzed and compared with the base case without the blind system, as studied by my colleagues J. Parra and A. Ravello in their senior project. The conclusion is that we can achieve significant energetic savings thanks to this design.

This 756-page book examines coal processing, surface forces and hydrophobicity, process improvements and environmental controls, dewatering and drying, gravity separations, industrial minerals flotation, base metal flotation, flotation equipment and practice, process reagents, magnetic and electrostatic separations, modeling and process control, and resource engineering.

This book collects the proceedings of the Parallel Computational Fluid Dynamics 2008 conference held in Lyon, France. Contributed papers by over 40 researchers representing the state of the art in parallel CFD and architecture from Asia, Europe, and North America examine major developments in (1) block-structured grid and boundary methods to simulate flows over moving bodies, (2) specific methods for optimization in Aerodynamics Design, (3) innovative parallel algorithms and numerical solvers, such as scalable algebraic multilevel preconditioners and the acceleration of iterative solutions, (4) software frameworks and component architectures for parallelism, (5) large scale computing and parallel efficiencies in the industrial context, (6) lattice Boltzmann and SPH methods, and (7) applications in the environment, biofluids, and nuclear engineering.

Crystallization is an important separation and purification process used in industries ranging from bulk commodity chemicals to specialty chemicals

and pharmaceuticals. In recent years, a number of environmental applications have also come to rely on crystallization in waste treatment and recycling processes. The authors provide an introduction to the field of newcomers and a reference to those involved in the various aspects of industrial crystallization. It is a complete volume covering all aspects of industrial crystallization, including material related to both fundamentals and applications. This new edition presents detailed material on crystallization of biomolecules, precipitation, impurity-crystal interactions, solubility, and design. Provides an ideal introduction for industrial crystallization newcomers Serves as a worthwhile reference to anyone involved in the field Covers all aspects of industrial crystallization in a single, complete volume Computer aided process engineering (CAPE) plays a key design and operations role in the process industries. This conference features presentations by CAPE specialists and addresses strategic planning, supply chain issues and the increasingly important area of sustainability audits. Experts collectively highlight the need for CAPE practitioners to embrace the three components of sustainable development: environmental, social and economic progress and the role of systematic and sophisticated CAPE tools in delivering these goals.
The book includes the best articles presented by researchers, academicians and industrial experts at

the International Conference on "Innovative Design and Development Practices in Aerospace and Automotive Engineering (I-DAD 2018)". The book discusses new concept in designs, and analysis and manufacturing technologies for improved performance through specific and/or multi-functional design aspects to optimise the system size, weight-to-strength ratio, fuel efficiency and operational capability. Other aspects of the conference address the ways and means of numerical analysis, simulation and additive manufacturing to accelerate the product development cycles.Describing innovative methods, the book provides valuable reference material for educational and research organizations, as well as industry, wanting to undertake challenging projects of design engineering and product development.

Advanced in fluid power engineering motion and control Power Transmission and Motion Control is a collection of papers showcased at the PTMC 2001 conference at the University of Bath. Representing the work of researchers and industry leaders from around the world, this book features the latest developments in power transmission, with an emphasis on motion and control studies from the field of fluid power engineering. Insight into current projects on the forefront of technology and innovation provides an overview of the current state of the field while informing ongoing work and

suggesting direction for future projects.

The book encompasses novel CFD techniques to compute offshore wind and tidal applications. Computational fluid dynamics (CFD) techniques are regarded as the main design tool to explore the new engineering challenges presented by offshore wind and tidal turbines for energy generation. The difficulty and costs of undertaking experimental tests in offshore environments have increased the interest in the field of CFD which is used to design appropriate turbines and blades, understand fluid flow physical phenomena associated with offshore environments, predict power production or characterise offshore environments, amongst other topics.

This book reports on topics at the interface between manufacturing, mechanical and chemical engineering. It gives special emphasis to CAD/CAE systems, information management systems, advanced numerical simulation methods and computational modeling techniques, and their use in product design, industrial process optimization and in the study of the properties of solids, structures, and fluids. Control theory, ICT for engineering education as well as ecological design, and food technologies are also among the topics discussed in the book. Based on the 2nd International Conference on Design, Simulation, Manufacturing: The Innovation Exchange (DSMIE-2019), held on June

11-14, 2019, in Lutsk, Ukraine, the book provides academics and professionals with a timely overview and extensive information on trends and technologies behind current and future developments of Industry 4.0, innovative design and renewable energy generation.

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.

This collection presents contributions on computational fluid dynamics (CFD) modeling and simulation of engineering processes from researchers and engineers involved in the modeling of multiscale and multiphase phenomena in material processing systems. The following processes are covered: Additive Manufacturing (Selective Laser Melting and Laser Powder Bed Fusion); Ironmaking and Steelmaking (Ladle Metallurgical Furnace, EAF, Continuous Casting, Blown Converter, Reheating Furnace, Rotary Hearth Furnace); Degassing; High Pressure Gas Atomization of Liquid Metals; Electroslag Remelting; Electrokinetic Deposition; Friction Stir Welding; Quenching; High Pressure Die Casting; Core Injection Molding; Evaporation of Metals; Investment Casting; Electromagnetic Levitation; Ingot Casting; Casting and Solidification with External Field (electromagnetic stirring and

ultrasonic cavitation) Interaction and Microstructure Evolution The collection also covers applications of CFD to engineering processes, and demonstrates how CFD can help scientists and engineers to better understand the fundamentals of engineering processes.

There are many sources of emissions produced by burning fuel for power or heat, through chemical reactions, and from leaks from industrial processes or equipment. There is always a possibility of a potential hazard when these gases enter into the indoor environment with the air flow. The determination of the concentration profiles are necessary to evaluate the potential hazard posed by the gas spread. The main objectives of this work are to develop an appropriate measurement methodology and a 3D CFD transient multicomponent simulation model for the determination of spatial and temporal distribution of gaseous emissions with the air flow in the indoor environment. This work is also aimed at comparing the numerical simulation results of different CFD programs for a 2D base case model of indoor air flow with and without emission source under laminar and turbulent flow conditions for the purpose of developing a better basic understanding of the physical phenomena and for the selection of the suitable and appropriate CFD program for the further development of the simulation model. One of the

goals is also to apply the developed simulation model to the loss prevention and risk mitigation in the indoor environment and to study the influence of different parameters on the concentration distribution of gaseous pollutants in the presence of air flow in the indoor environment to minimize the expensive and time consuming experimentation efforts.
The 24th International Symposium on Shock Waves (ISSW24) was held at the Beijing Friendship Hotel during July 11-16, 2004, in Beijing. It was a great pleasure for the Local Organizing Committee to organize the ISSW in China for the first time, because forty-seven years have passed since the First Shock Tube Symposium was held in 1957 at Albuquerque. The ISSW24 had to be postponed for one year because of the SARS outbreak in Beijing shortly before the Symposium was scheduled to be held in 2003, but it has achieved success due to the continuous support and kind understanding from all the delegates. It is very heart-warming to have had such an experience and I am very happy to have served as chairman for the Symposium. I would like to thank all for the contributions and help that they have given us over the past three years, without which we would not have had the Symposium. A total of 460 abstracts were submitted to the ISSW24. Each of the abstracts was evaluated by three members of the Scientific Review Committee and the decision on acceptance wasmade based on the

reviewers' reports. 195oral papers,including 9plenary lectures, wereaccepted to be presented in three parallel sessions, and 135poster papers in three dedicated poster sessions. Topics discussed in these papers cover all aspects ofshock wave research.

Discusses the CFD-DEM method of modeling which combines both the Discrete Element Method and Computational Fluid Dynamics to simulate fluid-particle interactions. Deals with both theoretical and practical concepts of CFD-DEM, its numerical implementation accompanied by a hands-on numerical code in FORTRAN Gives examples of industrial applications

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

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.

This book is intended to serve as a reference text for advanced scientists and research engineers to solve a variety of fluid flow problems using computational fluid dynamics (CFD). Each chapter arises from a collection of research papers and discussions contributed by the practiced experts in the field of fluid mechanics. This material has encompassed a wide range of CFD applications concerning computational scheme, turbulence modeling and its simulation, multiphase flow modeling, unsteady-flow computation, and industrial applications of CFD.

Copyright: 0547bb76d14c5793577727192e359e09