# Cfd Simulations Of Pollutant Gas Dispersion With Different

This book discusses energy transfer, fluid flow and pollution in built environments. It provides a comprehensive overview of the highly detailed fundamental theories as well as the technologies used and the application of heat and mass transfer and fluid flow in built environments, with a focus on the mathematical models and computational and experimental methods. It is a valuable resource for researchers in the fields of buildings and environment, heat transfer and global warming.

Next generation gas turbines will be required to produce low concentrations of pollutants such as oxides of nitrogen (NOx), carbon monoxide (CO), and soot. In order to design gas turbines which produce lower emissions it is essential to have computational tools to help designers. Over the past few decades, computational fluid dynamics (CFD) has played a key role in the design of turbomachinary and will be heavily relied upon for the design of future components. In order to design components with the least amount of experimental rig testing, the ensemble of submodels used in simulations must be known to accurately predict the component's performance. The present work aims to validate a CFD model used for a reverse flow, rich-burn, quick quench, lean-burn combustor being developed at Honeywell. Initially, simulations are performed to establish a baseline which will help to assess impact to combustor performance made by changing CFD models. Rig test data from Honeywell is compared to these baseline simulation results. Reynolds averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) turbulence models are both used with the presumption that the LES turbulence model will better predict combustor performance. One specific model, the fuel spray model, is evaluated next. Experimental data of the fuel spray in an isolated environment is used to evaluate models for the fuel spray and a new, simpler approach for inputting the spray boundary conditions (BC) in the combustor is developed. The combustor is simulated once more to evaluate changes from the new fuel spray boundary conditions. This CFD model is then used in a predictive simulation of eight other combustor configurations. All computer simulations in this work were preformed with the commercial CFD software ANSYS FLUENT. NOx pollutant emissions are predicted reasonably well across the range of configurations tested using the RANS turbulence model. However, in LES, significant under predictions are seen. Causes of the under prediction in NOx concentrations are investigated. Temperature metrics at the exit of the combustor, however, are seen to be better predicted with LES.

In the context of urbanization and compact urban living, conventional experience-based planning and design often cannot adequately address the serious environmental issues, such as thermal comfort and air quality. The ultimate goal of this book is to facilitate a paradigm shift from the conventional experience-based ways to a more scientific, evidence-based process of decision making in both urban planning and architectural design stage. This book introduces novel yet practical modelling and mapping methods, and provides scientific understandings of the urban typologies and wind environment from the urban to building scale through real examples and case studies. The tools provided in this book aid a systematic implementation of environmental information from urban planning to building design by making wind

information more accessible to both urban planners and architects, and significantly increasing the impact of urban climate information on the practical urban planning and design. This book is a useful reference book to architectural postgraduates, design practitioners and planners, urban climate researchers, as well as policy makers for developing future livable and sustainable cities.

This volume gathers the latest advances, innovations, and applications in the field of mining, geology and geo-spatial technologies, as presented by leading researchers and engineers at the International Conference on Innovations for Sustainable and Responsible Mining (ISRM), held in Hanoi, Vietnam on October 15-17 2020. The contributions cover a diverse range of topics, including mining technology, drilling and blasting engineering, tunneling and geotechnical applications, mineral processing, mine management and economy, environmental risk assessment and management, mining and local development, mined land rehabilitation, water management and hydrogeology, regional Geology and tectonics, spatial engineering for monitoring natural resources and environment change, GIS and remote sensing for natural disaster monitoring, risk mapping and revisualization, natural resources monitoring and management, mine occupational safety and health. Selected by means of a rigorous peer-review process, they will spur novel research directions and foster future multidisciplinary collaborations.

These conference proceedings provide a comprehensive overview of and in-depth technical information on all possible bioenergy resources (solid, liquid, and gaseous), including cutting-edge themes such as advanced fuels and biogas. The book includes current state-of-the-art topics ranging from feedstocks and cost-effective conversion processes to biofuels economic analysis and environmental policy, and features case studies and quizzes for each section derived from the implementation of actual hands-on biofuel projects to aid learning. It offers readers a starting point on this challenging and exciting path. The central concepts are defined and explained in the context of process applications under various topics. By focussing on the pertinent fundamental principles in the environment and energy sciences and by repeatedly emphasizing the importance of their correlation, it offers a strong foundation for future study and practice. Learning about fundamental properties and mechanisms on an ongoing basis is absolutely essential for long-term professional viability in a technically vibrant area such as nanotechnology. The book has been written for undergraduate and graduate students in chemical, energy and environment engineering. However, selected sections can provide the basis for courses in civil, mechanical or electrical engineering. It includes a self-contained presentation of the key concepts of energy resources, solar thermal and photovoltaic systems, nuclear energy, biomass conversion technology and agricultural-waste processing. Throughout it interweaves descriptive material on sustainable development, clean coal technology, green technology, solid-waste management and lifecycle assessments. It offers an introduction to these topics rather than comprehensive coverage of the themes and their in-depth fundamentals.

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 volume presents the proceedings of the International Conference on Medical and Biological Engineering held from 16 to 18 March 2017 in Sarajevo, Bosnia and Herzegovina. Focusing on the theme of ‘Pursuing innovation. Shaping the future’, it highlights the latest advancements in Biomedical Engineering and also presents the latest findings, innovative solutions and emerging challenges in this field. Topics include: - Biomedical Signal Processing - Biomedical Imaging and Image Processing - Biosensors and Bioinstrumentation - Bio-Micro/Nano Technologies - Biomaterials - Biomechanics, Robotics and Minimally Invasive Surgery - Cardiovascular, Respiratory and Endocrine Systems Engineering - Neural and Rehabilitation Engineering - Molecular, Cellular and Tissue Engineering - Bioinformatics and Computational Biology - Clinical Engineering and Health Technology Assessment - Health Informatics, E-Health and Telemedicine - Biomedical Engineering Education - Pharmaceutical Engineering

In recent decades, the field of computational fluid dynamics has made significant advances in enabling advanced computing architectures to understand many phenomena in biological, geophysical, and engineering fluid flows. Almost all research areas in fluids use numerical methods at various complexities: from molecular to continuum descriptions; from laminar to turbulent regimes; from low speed to hypersonic, from stencil-based computations to meshless approaches; from local basis functions to global expansions, as well as from first-order approximation to high-order with spectral accuracy. Many successful efforts have been put forth in dynamic adaptation strategies, e.g., adaptive mesh refinement and multiresolution representation approaches. Furthermore, with recent advances in artificial intelligence and heterogeneous computing, the broader fluids community has gained the momentum to revisit and investigate such practices. This Special Issue, containing a collection of 13 papers, brings together researchers to address recent numerical advances in 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.

This book gathers the proceedings of the 8th International Symposium on Coal Combustion. The contributions reflect the latest research on coal quality and combustion, techniques for pulverized coal combustion and fluidized bed combustion, special issues regarding CO2 capture (CCS), industrial applications, etc. – aspects that are of great importance in promoting academic communications between related areas and the technical development of coal-related fields. The International Symposium on Coal Combustion (ISCC), sponsored and organized by Tsinghua University since 1987, has established itself as an important platform allowing scientists and engineers to exchange information and ideas on the science and technology of coal combustion and related issues, and to forge new partnerships in the growing Chinese market. Researchers in the fields of clean coal combustion, carbon dioxide capture and storage, coal chemical engineering, energy engineering, etc. will greatly benefit from this book. Guangxi Yue, professor of the Department of Thermal Engineering in Tsinghua University, Beijing, China, and a member of Chinese Academy of Engineering(CAE). Shuiqing Li, professor of the Department of Thermal Engineering in Tsinghua University, Beijing, China.

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.

"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.

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

Human exposure to gas-phase pollutants can lead to adverse health effects such as respiratory, cardiovascular, and neurological diseases. When some gases undergo chemical reactions in buildings, a number of reaction products are created and can influence the health and productivity of occupants. These air pollutants are

heterogeneously distributed by air flow patterns, surface deposition, and chemical reactions so that the health effects of these species are different from one space to another in indoor environments. However, very little information is available in the literature on how the indoor environmental conditions affect concentrations and spatial distributions of reactants and reaction products. Based on this background, the objective of this study is to examine chemical processes and pollutant dynamics of gas-phase compounds in indoor environments under representative conditions of chemical reactions, ventilation, lighting, and indoor surfaces. This Ph.D. dissertation used three computational fluid dynamics (CFD) model frameworks designed to investigate spatial distributions of gas-phase species. The focus involved three critical reactive species (ozone, hydroxyl radical (OH), and chlorine atom (Cl)) in indoor environments as they can dominate indoor oxidation processes and reactive chemistry. Three different reaction scenarios were simulated: 1) ozone interaction with human surfaces; 2) indoor photolysis of nitrous acid (HONO); and 3) indoor surface cleaning by a chlorine bleach solution. The first modeling study showed that ozone was depleted on the human surface due to ozone reactions with skin oil and soiled clothing. Because of the ozone surface reaction, primary products were relatively concentrated near occupants, while secondary products were relatively well distributed throughout the room. Clean clothing with lower amounts of skin oil produced about 40% lower primary reaction products than the soiled clothing condition. Increasing air mixing near the human surface also enhanced ozone uptake to the human surface. With regard to indoor photolysis of HONO, production and spatial distribution of reaction products in indoor environments vary highly with light conditions. Photolysis of HONO generated OH that led to recycling reactions between OH and the hydroperoxy radical (HO2). Due to their high reactivities, such radicals (OH and HO2) are mainly concentrated where they are generated, while the oxidation products were produced in the lighting zone and dispersed to the ambient air. The increased volume of a daylight zone produced more oxidation products. Artificial lights also photolyzed HONO, but the impact was marginal compared to direct sunlight, due to the intensity decrease with increasing distance from the light source. The third CFD model simulated bleach cleaning experiments conducted in a test house where bleach solutions were applied to the living room floor. From the cleaning surface, hypochlorous acid (HOCl) and nitryl chloride (ClNO2) were emitted. Uptake of HOCl to aerosol surfaces produced chlorine (Cl2). These three gas-phase species (i.e., HOCl, ClNO2, and Cl2) were removed at approximately 80% due to surface deposition and 20% by ventilation. Photolysis of HOCl, ClNO2, and Cl2 were the key processes that generated radicals (OH and Cl). The radicals were confined in the sunlit zone and produced some toxic gas-phase species such as hydrogen chloride (HCl). Once oxidation products were generated, they were dispersed and recirculated to the ambient air by an indoor ventilation system. However, even with a high air mixing rate (8 h-1) and all indoor doors opened, the concentrations of the cleaning products in the bleach cleaning zone were 2-3 times higher than those in other rooms. In addition, regardless of ventilation conditions, the reactive species (OH and Cl) were concentrated near their sources, mainly due to the reaction time scale that was notably shorter than the transport time scale.

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.

Industrial Ventilation Design Guidebook, Volume 2: Engineering Design and Applications brings together researchers, engineers (both design and plants), and scientists to develop a fundamental scientific understanding of ventilation to help engineers implement state-of-the-art ventilation and contaminant control technology. Now in two volumes, this reference contains extensive revisions and updates as well as a unique section on best practices for the following industrial sectors: Automotive; Cement; Biomass Gasifiers; Advanced Manufacturing; Industrial 4.0); Non-ferrous Smelters; Lime Kilns; Pulp and Paper; Semiconductor Industry; Steelmaking; Mining. Brings together global researchers and engineers to solve complex ventilation and contaminant control problems using state-of-the-art design equations Includes an expanded section on modeling and its practical applications based on recent advances in research Features a new chapter on best practices for specific industrial sectors

Combustion technology has traditionally been dominated by air/fuel combustion. However, two developments have increased the significance of oxygen-enhanced combustion—new technologies that produce oxygen less expensively and the increased importance of environmental regulations. Advantages of oxygen-enhanced combustion include less pollutant emissions as well as increased energy efficiency and productivity. Oxygen-Enhanced Combustion, Second Edition compiles information about using oxygen to enhance industrial heating and melting processes. It integrates fundamental principles, applications, and equipment design in one volume, making it a unique resource for specialists implementing the use of oxygen in combustion systems. This second edition of the bestselling book has more than doubled in size. Extensively updated and expanded, it covers significant advances in the technology that have occurred since the publication of the first edition. What’s New in This Edition Expanded from 11 chapters to 30, with most of the existing chapters revised A broader view of oxygen-enhanced combustion, with more than 50 contributors from over 20 organizations around the world More coverage of fundamentals, including fluid flow, heat transfer, noise, flame impingement, CFD modeling, soot formation, burner design, and burner testing New chapters on applications such as flameless combustion, steel reheating, iron production, cement production, power generation, fluidized bed combustion, chemicals and petrochemicals, and diesel engines This book offers a unified, up-to-date look at important commercialized uses of oxygen-enhanced combustion in a wide range of industries. It brings together the latest knowledge to assist those researching, engineering, and implementing combustion in power plants,

engines, and other applications.

This book constitutes the refereed proceedings of the 25th International Conference on Parallel Computational Fluid Dynamics, ParCFD 2013, held in Changsha, China, in May 2013. The 35 revised full papers presented were carefully reviewed and selected from more than 240 submissions. The papers address issues such as parallel algorithms, developments in software tools and environments, unstructured adaptive mesh applications, industrial applications, atmospheric and oceanic global simulation, interdisciplinary applications and evaluation of computer architectures and software environments.

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.

Good,No Highlights,No Markup,all pages are intact, Slight Shelfwear,may have the corners slightly dented, may have slight color changes/slightly damaged spine.

The air distribution in occupied spaces is a major issue of public concern. It is widely recognized that the quality of air and the nature of airflow can affect the health of occupants and the energy consumed in buildings and transport vehicles. ROOMVENT is the principal international conference in the field of air distribution. It was first initiated in 1987 by SCANVAC, the Scandinavian Federation of Heating, Ventilating and Sanitary Engineering Associations in Denmark, Finland, Iceland, Norway and Sweden. The aim of the Conference is to bring together researchers from universities and research institutes, engineers from industry and government officials and policy makers, with the goal of experiencing the latest techniques for measuring and analyzing indoor air flow, the visualization of indoor air flow patterns, the evaluation of ventilation parameters and the most recent developments in computer simulation techniques of room airflow. It is hoped that the theme of ROOMVENT 2000 "Ventilation for Health and Sustainable Environment" will set the scene for room air distribution research and development for the new millennium.

The development of clean, sustainable energy systems is one of the pre-eminent issues of our time. Most projections indicate that combustion-based energy conversion systems will continue to be the predominant approach for the majority of our energy usage, and gas turbines will continue to be important combustion-based energy conversion devices for many decades to come, used for aircraft propulsion, ground-based power generation, and mechanical-drive applications. This book compiles the key scientific and technological knowledge associated with gas turbine emissions into a single authoritative source. The book has three sections: the first section reviews major issues with gas turbine combustion, including design approaches and constraints, within the context of emissions. The second section addresses fundamental issues associated with pollutant formation, modeling, and prediction. The third section features case studies from manufacturers and technology developers, emphasizing the system-level and

practical issues that must be addressed in developing different types of gas turbines that emit pollutants at acceptable levels.

The book presents a state-of-the-art in environmental aerodynamics and the structural design of wind energy support structures, particularly from a modern computational perspective. Examples include real-life applications dealing with pollutant dispersion in the building environment, pedestrian-level winds, comfort levels, relevant legislation and remedial measures. Design methodologies for wind energy structures include reliability assessment and code frameworks.

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 numerical simulation of combustion processes in internal combustion engines, including also the formation of pollutants, has become increasingly important in the recent years, and today the simulation of those processes has already become an indispensable tool when - veloping new combustion concepts. While pure thermodynamic models are well-established tools that are in use for the simulation of the transient behavior of complex systems for a long time, the phenomenological models have become more important in the recent years and have also been implemented in these simulation programs. In contrast to this, the thr- dimensional simulation of in-cylinder combustion, i. e. the detailed, integrated and continuous simulation of the process chain injection, mixture formation, ignition, heat release due to combustion and formation of pollutants, has been significantly improved, but there is still a number of challenging problems to solve, regarding for example the exact description of s- processes like the structure of turbulence during combustion as well as the appropriate choice of the numerical grid. While chapter 2 includes a short introduction of functionality and operating modes of internal combustion engines, the basics of

kinetic reactions are presented in chapter 3. In chapter 4 the physical and chemical processes taking place in the combustion chamber are described. Chter 5 is about phenomenological multi-zone models, and in chapter 6 the formation of poll- ants is described.

This book considers the pollutants formed by the combustion of solid biomass fuels. The availability and potential use of solid biofuels is first discussed because this is the key to the development of biomass as a source of energy. This is followed by details of the methods used for characterisation of biomass and their classification. The various steps in the combustion mechanisms are given together with a compilation of the kinetic data. The chemical mechanisms for the formation of the pollutants: NOx, smoke and unburned hydrocarbons, SOx, Cl compounds, and particulate metal aerosols are given in detail. Combustion kinetics required for the application for design purposes are given. Examples are given of emission levels of a range different types of combustion equipment. Data is given of NOx, particulates and other pollutant arising from combustion of different fuels in fixed bed combustion, fluidized bed combustion and pulverised biomass combustion and co-firing. Modeling methods including computational fluid dynamics for the various pollutants are outlined. The consequential issues arising from the wide scale use of biomass and future trends are then discussed. In particular the role of carbon capture and storage in large biomass combustion plants is considered as well as the opportunity of reducing the concentration of atmospheric concentration of carbon dioxide.

Until now, anyone conducting industrial combustion tests had to either rely on old methods, go scurrying through the literature to find proven applicable methodologies, or hire top-shelf consultants such as those that work for cutting-edge companies like John Zink. Manufacturers can no longer take industrial combustion for granted. Air and noise po

This book contains twelve chapters detailing significant advances and applications in fluid dynamics modeling with focus on biomedical, bioengineering, chemical, civil and environmental engineering, aeronautics, astronautics, and automotive. We hope this book can be a useful resource to scientists and engineers who are interested in fundamentals and applications of fluid dynamics.

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

Emissions from civil subsonic air traffic are potentially in sufficient amounts to affect climate, particularly with the projected growth in air traffic. These six papers from a November 1999 seminar present advances in new low-emission and fuel efficient technologies aimed at addressing environmental concerns in a balanced manner. The topics are legislation and control technology, improved combustor turndown using a variable swirl number fuel injector, the aero engine response to the protection of the global atmosphere, predictive methods for gas turbine combustor emissions, the influence of premixture equivalence ratio on the thermo-acoustic response of a swirl burner, and the modeling of CO and NO formation in turbulent jet flames. No index. Distributed by ASME. c. Book News Inc.

This book offers a collection of original peer-reviewed contributions presented at the 7th International Congress on Design and Modeling of Mechanical Systems (CMSM'2017), held in Hammamet, Tunisia, from the 27th to the 29th of March 2017. It reports on both research findings, innovative industrial applications and case studies concerning mechanical systems and related to modeling and analysis of materials and structures, multiphysics methods, nonlinear dynamics, fluid structure interaction and vibroacoustics, design and manufacturing engineering. Continuing on the tradition of the previous editions, this proceedings offers a broad overview on the state-of-the art in the field and a useful resource for academic and industry specialists active in the field of design and modeling of mechanical systems. CMSM'2017 was jointly organized by two leading Tunisian research laboratories: the Mechanical, Modeling and Manufacturing Laboratory of the National Engineering School of Sfax and the Mechanical Engineering Laboratory of the National Engineering School of Monastir..

Advances in Oxygen Research and Application: 2013 Edition is a ScholarlyBrief™ that delivers timely, authoritative, comprehensive, and specialized information about ZZZAdditional Research in a concise format. The editors have built Advances in Oxygen Research and Application: 2013 Edition on the vast information databases of ScholarlyNews.™ You can expect the information about ZZZAdditional Research in this book to be deeper than what you can access anywhere else, as well as consistently reliable, authoritative, informed, and relevant. The content of Advances in Oxygen Research and Application: 2013 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/.

Public health studies have indicated strong correlations between air pollution and adverse respiratory and cardiovascular effects, premature death, and mortality. Widely used in regulatory applications for permitting new sources and in health studies to estimate exposure, Gaussian-based atmospheric dispersion models play a crucial role in air quality management but have several well-documented limitations. The main challenge of improving dispersion models is how to represent the complex turbulent flow field from the source(s) to the receptor(s) at a local length scale (~ 1 km) with high resolutions (~ 1 m). At the local scale, built environment and/or moving vehicle induced turbulence have significant effects on the dispersion process. This dissertation presents the Computational Fluid Dynamics (CFD) modeling of the turbulent, reactive flow fields, the dispersion, and transformation of the air pollutants from two major air pollution sources, the power generation facilities and the highway vehicles. These two major sources could be close to communities and correspondingly raise

local air quality concerns. For example, the hydrocarbon fueled distributed generation (DG) facilities are usually located near end-users and have shorter stacks than those of centralized power plants. Communities near major highways are exposed to elevated air pollutant concentrations than those far away from highways. The methods proposed in this dissertation would improve dispersion models and thereby public health research. For the power generators, a CFD-aided air pollutant dispersion parameterization method was developed, and it showed better performance than a regulatory dispersion model. This work was based on studies of a centralized power plant (1,235 MW), a simple cycled gas turbine (47 MW) with a heat recovery system, and a combined heat and power gas turbine (CHP, two units, 15 MW each). For the highway vehicles, NO2/NOx ratios at different plume evolution stages were clearly defined. Curbside measurement, on-road chasing measurement, and CFD simulations demonstrated on-road NO2 formation could be a large contributor of the total NO2 at the curbside, which was neglected in the current dispersion models. A CFD-aided on-road and near-road NO2/NOx ratio parameterization method was proposed and evaluated using curbside measurement data collected at an interstate highway.