Cfd Simulations Of Pollutant Gas Dispersion With Different

MILD Combustion: Modelling Challenges, Experimental Configurations and Diagnostic Tools
Pollutant Dispersion in Built Environment
Advances in Modeling of Fluid Dynamics
Risk, Reliability and Safety: Innovating Theory and Practice
Electrostatic Precipitation
Advances in Oxygen Research and Application: 2013 Edition
IoT for Smart Operations in the Oil and Gas Industry
Investigation of Pollutants Inter-unit Dispersion and Natural Ventilation in Buildings Due to Wind Effect
Energy 2000
Combustion
Advances in Modeling of Fluid Dynamics
Proceedings of the International Conference on Innovations for Sustainable and Responsible Mining
ECOS 2012 The 25th International Conference on Efficiency, Cost, Optimization and Simulation of Energy Conversion Systems and Processes (Perugia, June 26th-June 29th, 2012)
Air Pollution from Motor Vehicles
Large Eddy Simulations of a Reverse Flow Combustion System
La Modélisation multidimensionnelle des écoulements dans les moteurs
Pollutants Generated by the Combustion of Solid Biomass Fuels
Gas Turbine Emissions
Paper
Gas Turbine Pollutant Emissions
A New Framework for Understanding Indoor Chemical Processes and Dynamics Using Computational Fluid Dynamics (CFD) Simulations
Multiscale Modeling for Process Safety Applications

Simulating Combustion
Urban Wind Environment
Numerical and experimental investigations of distribution of gaseous emissions with the air flow in the indoor environment
Scientific and Technical Aerospace Reports
Comparison Between Experiments and CFD Predictions of MixedConvection Flows in an Atrium
Physics-based, Data-driven Modeling of Micro-environmental Air Quaity Impact from Stationary and Mobile Sources
Handbook of Fire and the Environment
Cleaner Treatment Technologies and Productions in The Energy Industry
Clean Coal Technology and Sustainable Development
Air Distribution in Rooms
Recent Numerical Advances in Fluid Mechanics
Design and Modeling of Mechanical Systems—III
CFD Modeling and Simulation in Materials Processing 2018
The Finite Volume Method in Computational Fluid Dynamics
Parallel Computational Fluid Dynamics
Computational Fluid Dynamics (CFD) of Chemical Processes
Computational Fluid Dynamics Simulations

GUADALUPE BRYCE

MILD Combustion: Modelling Challenges, Experimental Configurations and Diagnostic Tools Springer
"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.
Pollutant Dispersion in Built Environment Springer Nature
Contributions by Surhid Gautam and Lit-Mian Chan. This book presents a state-of-the art review of vehicle emission standards and regulations and provides a synthesis of worldwide experience with vehicle emission control technologies and their applications in both industrial and developing countries. Topics covered

include: * The two principal international systems of vehicle emission standards: those of North America and Europe * Test procedures used to verify compliance with emissions standards and to estimate actual emissions * Engine and aftertreatment technologies that have been developed to enable new vehicles to comply with emission standards, as well as the cost and other impacts of these technologies * An evaluation of measures for controlling emissions from in-use vehicles * The role of fuels in reducing vehicle emissions, the benefits that could be gained by reformulating conventional gasoline and diesel fuels, the potential benefits of alternative cleaner fuels, and the prospects for using hydrogen and electric power to run motor vehicles with ultra-low or zero emissions. This book is the first in a series of publications on vehicle-related pollution and control measures prepared by the World Bank in collaboration with the United Nations Environment Programme to underpin the Bank's overall objective of promoting transport that is environmentally sustainable and least damaging to human health and welfare.

Advances in Modeling of Fluid Dynamics CRC Press

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.

Risk, Reliability and Safety: Innovating Theory and Practice Springer Science & Business Media

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.

Electrostatic Precipitation Springer

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/.

Advances in Oxygen Research and Application: 2013 Edition Springer

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.

Frontiers Media SA

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.

IoT for Smart Operations in the Oil and Gas Industry Editions TECHNIP

The 8-volume set contains the Proceedings of the 25th ECOS 2012 International Conference, Perugia, Italy, June 26th to June 29th, 2012. ECOS is an acronym for Efficiency, Cost, Optimization and Simulation (of energy conversion systems and processes), summarizing the topics covered in ECOS: Thermodynamics, Heat and Mass Transfer, Exergy and Second Law Analysis, Process Integration and Heat Exchanger Networks, Fluid Dynamics and Power Plant Components, Fuel Cells, Simulation of Energy Conversion Systems, Renewable Energies, Thermo-Economic Analysis and Optimisation, Combustion, Chemical Reactors, Carbon Capture and Sequestration, Building/Urban/Complex Energy Systems, Water Desalination and Use of Water Resources, Energy Systems- Environmental and Sustainability Issues, System Operation/ Control/Diagnosis and Prognosis, Industrial Ecology.

Investigation of Pollutants Inter-unit Dispersion and Natural Ventilation in Buildings Due to Wind Effect CRC Press

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.

Energy 2000 Springer Nature

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..

Combustion Elsevier

This book provides a rigorous treatment of the coupling of chemical reactions and fluid flow. Combustion-specific topics of chemistry and fluid mechanics are considered and tools described for the simulation of combustion processes. This edition is completely restructured. Mathematical Formulae and derivations as well as the space-consuming reaction mechanisms have been replaced from the text to appendix. A new chapter discusses the impact of combustion processes on the atmosphere, the chapter on auto-ignition is extended to combustion in Otto- and Diesel-engines, and the chapters on heterogeneous combustion and on soot formation are heavily revised.

Advances in Modeling of Fluid Dynamics Butterworth-Heinemann

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.

Proceedings of the International Conference on Innovations for Sustainable and Responsible Mining Springer

With an increasingly challenging commercial environment, and the need imposed by safety principles to reduce both fuel consumption and pollutant emissions, the development of new engines can now benefit from the advances of computational fluid dynamics. Engine CFD is a most challenging simulation problem. This is caused by the spread of time and space scales, the excursion amplitude of most parameters, the high quasi-cyclic unstationarity of engine flows, the importance of minor geometry details, the number of physical and chemical processes including turbulent combustion and multi-phase flows to model.

However, engine CFD has now reached a state where it has become a widely used tool, not only for engine understanding, but also increasingly for engine design. Undoubtedly, laser diagnostics in optical access engines have also brought significant help.Contents: 1. State of the art of multi-dimensional modeling of engine reacting flows. 2. Simulation of the intake and compression strokes of a motored 4-valve SI engine with a finite element code. 3. A parallel, unstructured-mesh methodology for device-scale combustion calculations. 4. Large-eddy simulation of in-cylinder flows. 5. Simulation of engine internal flows using digital physics. 6. Automatic block decomposition of parametrically changing volumes. 7. Developments in spray modeling in diesel and direct-injection gasoline engines. 8. Cyto-fluid dynamic theory of atomization processes. 9. Influence of the wall temperature on the mixture preparation in DI gasoline engines. 10. Simulation of cavitating flows in diesel injectors. 11. Recent developments in simulations of internal flows in high pressure swirl injectors. 12. 3D simulation of DI diesel combustion and pollutant formation using a two-component reference fuel. 13. Modeling of NOx and soot formation in diesel combustion. 14. Multi-dimensional modeling of combustion and pollutants formation of new technology light duty diesel engines. 15. 3D modeling of combustion for DI-SI engines. 16. Combustion modeling with the G-equation. 17. Multi-dimensional modeling of the aerodynamic and combustion in diesel engines. 18. CFD aided development of a SI-DI engine. 19. CFD engine applications at FIAT research centre. 20. Application of a detailed emission model for heavy duty diesel engine simulations. 21. CFD based shape optimization of IC engine.

ECOS 2012 The 25th International Conference on Efficiency, Cost, Optimization and Simulation of Energy Conversion Systems and Processes (Perugia, June 26th-June 29th, 2012) Springer

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.

Air Pollution from Motor Vehicles Elsevier

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.

Large Eddy Simulations of a Reverse Flow Combustion System Firenze University Press

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.

La Modélisation multidimensionnelle des écoulements dans les moteurs Springer

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.

Pollutants Generated by the Combustion of Solid Biomass Fuels World Bank Publications

Risk, Reliability and Safety contains papers describing innovations in theory and practice contributed to the scientific programme of the European Safety and Reliability conference (ESREL 2016), held at the University of Strathclyde in Glasgow, Scotland (25—29 September 2016). Authors include scientists, academics, practitioners, regulators and other key individuals with expertise and experience relevant to specific areas. Papers include domain specific applications as well as general modelling methods. Papers cover evaluation of contemporary solutions, exploration of future challenges, and exposition of concepts, methods and processes. Topics include human factors, occupational health and safety, dynamic and systems reliability modelling, maintenance optimisation, uncertainty analysis, resilience assessment, risk and crisis management.

Gas Turbine Emissions MDPI

CFD simulations were conducted to evaluate the influence of an upstream building on the inter-unit dispersion and natural ventilation performance in the downstream interfered multistory buildings. A tracer gas was employed to simulate gaseous and fine particle pollutants. The presence of an upstream building greatly changes the airflow field and pollutant transportation routes in and around the downstream interfered building. The influence of the height of the upstream building was also examined. Under the normal wind incidence, the low upstream building greatly increases the average indoor air change rate per hour (ACH) values and the pollutant re-entry ratios (Rk) below the source unit on the windward side of the downstream interfered building. Under the oblique wind, a high upstream building greatly increases the average ACH values on the windward side and increases the Rk on the leeward side of the downstream building. CFD simulations were also conducted to investigate the combined effect of surrounding buildings and envelope features on the inter-unit dispersion and natural ventilation performance in multi-story buildings. The effect of three typical envelope features including balconies with upper and lower vents, wing walls, and bay windows were investigated. The results show that the influence of envelope features on ventilation performance and inter-unit dispersion is highly dependent on the surrounding environment including the approaching wind directions, the orientation of the target building and the height of the upstream building. For an isolated building, the presence of envelope features, particularly the vertical wing walls, improves largely the natural ventilation performance of the building but enhances pollutants inter-unit dispersion. However, when considering the effect of an upstream building, the presence of envelope features mostly lowers the natural ventilation performance but weakens the inter-unit dispersion, since the near-wall airflow pattern and pressure distribution are totally redefined.

Paper ScholarlyEditions
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.