Numerical Simulation Of Particle Deposition In Cross Flow

Further progress in achieving the objectives of the project was made in the period of July 1 to September 30, 1997. The direct numerical simulation of particle removal process in turbulent gas flows was continued. Variations of vorticity contours which are averaged over a short time duration are studied. It is shown that the near wall vortices profoundly affect the particle removal process in turbulent boundary layer flows. The sublayer model for evaluating the particle deposition in turbulent flows was extended to include the effect of particle rebound. A new more advance flow model for the near wall vortices is also used in these analysis. Sample particle trajectories are obtained and discussed. Experimental data for transport and deposition of fibrous particles in the aerosol wind tunnel was obtained. The measured deposition velocity is compared with the empirical correlation and the available data and discussed. Particle resuspension process in turbulent flows are studied. The model is compared with the experimental data. It is shown that when the effects of the near wall flow structure, as well as the surface roughness are included the model agrees with the available experimental data.

Surface contamination is of cardinal importance in a host of technologies and industries, ranging from microelectronics to optics to automotive to biomedical. Thus, the need to understand the causes of surface contamination and their removal is very patent. Generally speaking, there are two broad categories of surface contaminants: film-type and particulates. In the world of shrinking dimensions, such as the ever-decreasing size of microelectronic devices, there is an intensified need to understand the behavior of nanoscale particles and to devise ways to remove them to an acceptable level. Particles which were functionally innocuous a few years ago are "killer defects" today, with serious implications for yield and reliability of the components. This book addresses the sources, detection, characterization and removal of both kinds of contaminants, as well as ways to prevent surfaces from being contaminated. A number of techniques to monitor the level of cleanliness are also discussed. Special emphasis is placed on the behavior of nanoscale particles. The book is amply referenced and profusely illustrated. • Excellent reference for a host of technologies and industries ranging from microelectronics to optics to automotive to biomedical. • A single source document addressing everything from the sources of contamination to their removal and prevention. • Amply referenced and profusely illustrated.
The transport of solid particles inside a laboratory-scale turbulent boundary-layer is studied by numerical simulations, to obtain a better understanding of the mechanisms associated with wind erosion of soil. The presence of one or several Gaussian hills allows a study of the topographic effects on the transport, deposition and re-emission of solid particles. The carrier fluid motion is resolved in a Large Eddy Simulation (LES). Wall models are implemented to better account for the effects of turbulent flow near the terrain. Particle trajectories are calculated using a Lagrangian tracking. Take-off and rebound models are developed in order to take into account particle emissions and impacts at the wall. In the first part, the flow over transversal Gaussian hills is simulated and validated by comparison with different experiments. According to Oke [1988], the flow inside an urban canopy can be schematically characterised into different flow regimes depending on the relative localisation of the obstacles at the ground. This concept is applied to the case of sand dunes, assimilated to 2D hills in this study. The focus is on the recirculation zone (RZ) on the lee side, which has the characteristic of increasing the residence time and the interaction fluid/particle in general, particle trapping and deposition in particular. The variations of RZ with different hill geometries and Reynolds numbers are examined. A study on the roughness sublayer is conducted in order to determine the roughness effects due to the layer of solid particles on the wall. The second part of the work is devoted to the simulation of solid particle transport over the Gaussian hills. The objective is to improve the modelling of particle take-off, rebound and the two-way coupling between the fluid and the particle. A first work of validation is conducted by using the complete model of solid particle transport developed in this thesis. In particular, the evolution of particle emission flux predicted by the take-off model is in accordance with classical saltation models and experiments from the literature. Over the Gaussian hills, analysis of particle transport is conducted using concentration and mean velocity fields. Two mappings are realised. The first indicates the intensity of the local and instantaneous flow structures that arguably regulate the re-entrainment of particles trapped inside the RZ. The second shows the accumulation of particles on the hills. These results highlight zones prone to wind erosion and particle deposition around the hills. Last but not least, the fluxes of particle trapping and deposition inside the RZ are quantified and compared to the incoming flux from upstream. These fluxes, albeit relatively weak in comparison to the incoming one, contribute potentially to dune migrations and desertification.

This volume continues previous DLES proceedings books, presenting modern developments in turbulent flow research. It is comprehensive in its coverage of numerical and modeling techniques for fluid mechanics. After Surrey in 1994, Grenoble in 1996, Cambridge in 1999, Enschede in 2001, Munich in 2003, Poitiers in 2005, and Trieste in 2009, the 8th workshop, DLES8, was held in Eindhoven, The Netherlands, again under the auspices of ERCOFTAC. Following the spirit of the series, the goal of this workshop is to establish a state-of-the-art of DNS and LES techniques for the computation and modeling of transitional/turbulent flows covering a broad scope of topics such as aerodynamics, acoustics, combustion, multiphase flows, environment, geophysics and biomedical applications. This gathering of specialists in the field was a unique opportunity for discussions about the more recent advances in the prediction, understanding and control of turbulent flows in academic or industrial situations.

The work presented in this monograph marks a new era, we believe, both in the development of quantitative anatomy of the lung, and in the correlation of anatomy with physiology. For many years, physiologists interested in the overall functioning of the lung have felt a need for better quantitative descriptions of pulmonary anatomy. As physiologists, we know a good deal about the forces operating to produce pulmonary ventilation, and the quantities that define this function in rest and exercise; and the same for effective distribution of air within the lung - "alveolar" ventilation; and for the exchange of respiratory gases between air and blood. There have been no correspondingly precise quantitative measurements of the pulmonary structures that serve these functions. The great advances in the study of pulmonary anatomy in the past decade have been chiefly in the realm of "fine structure". This has tended to bring together anatomy and biochemistry or physical chemistry, rather than anatomy and physiology. This conjunction has aided, for example, the conception of diffusion as a physicochemical process, but not that of diffusion as a metabolic bodily function. It was, therefore, a remarkably fortunate circumstance which brought together in our laboratory, about three years ago, Professor DoMINGO GoMEZ and Dr. EWALD R. WEIBEL: Professor GoMEZ a mathematician and biophysicist of distinction and long experience; Dr. WEIBEL a young anatomist trained under Pro fessor GIAN TöNDURY in Zürich, and with additional research experience with Professor A VERILL LIEBOW at Yale.

Cold spray is a rapidly developing coating technology for depositing materials in the solid state. The cold spray particle deposition process was simulated by modeling the high velocity impacts of spherical particles onto a flat substrate under various conditions. We, for the first time, proposed...
the Couple Eulerian Lagrangian (CEL) numerical approach to solve the high strain rate deformation problem. The capability of the CEL numerical approach in modeling the Cold Spray deposition process was verified through a systematic parameter study, including impact velocity, initial particle temperature, friction coefficient and materials combination. The simulation results by using the CEL numerical approach agree with the experimental results published in the literature. Comparing with other numerical approaches, which are Lagrangian, ALE and SPH, the CEL analyses are generally more accurate and more robust in higher deformation regimes. Besides simulating the single particle impact problem, we also extended our study into the simulation of multiple impacts. A FCC-like particles arrangement model that inspired by the crystal structure was built to investigate the porosity rate and residual stress of deposited particles under various conditions. We observed not only the 3D profiles of voids, but also their distributions and developments during different procedures. Higher impact velocity and higher initial temperature of particles are both of benefit to produce a denser cold spray coating. The compressive residual stresses existed in the interface between the particle and substrate is mainly caused by the large and fast plastic deformation. Another simplified model for multiple impacts was created for the simulation of surface erosion. A severe surface erosion is the result of a high impact velocity, a high friction coefficient and a low contact angle. Two element failure models suitable for high-strain-rate dynamic problems were introduced in this study. For a ductile material as Copper, it followed two fracture modes in our study, which are tensile failure mode and shear failure mode. The former one mainly occurred beneath the substrate surface and the periphery of substrate craters, nevertheless the latter one was found predominately at the surface of craters. Four steps were found during the propagation of crack: void formation; crack formation; crack growth; coalescence and failure. A simple criterion equation was derived based on the simulation results for predicting the initiation of damage, which the erosion velocity $v_{\text{ero}}$ is a function of contact angle and erosion velocity for normal impact $v_{\text{pi/2}}$. The equivalent plastic strain could also be a parameter for identifying the onset of damage, identified as being 1.042 for Copper in our study.

Particle-laden turbulent flow through a straight square duct at $Re_{\text{tau}}=300$ is studied using direct numerical simulation (DNS) and Lagrangian particle tracking. A parallelized 3-D particle tracking direct numerical simulation code has been developed to perform the large-scale turbulent particle transport computations reported in this thesis. The DNS code is validated after demonstrating good agreement with the published DNS results for the same flow and Reynolds number. Lagrangian particle transport computations are carried out using a large ensemble of passive tracers and finite-inertia particles and the assumption of one-way fluid-particle coupling. Using four different types of initial particle distributions, Lagrangian particle dispersion, concentration and deposition are studied in the turbulent straight square duct. Particles are released in a uniform distribution on a cross-sectional plane at the duct inlet, released as particle pairs in the core region of the duct, distributed randomly in the domain or distributed uniformly in planes at certain heights above the walls. One- and two-particle dispersion statistics are computed and discussed for the low Reynolds number inhomogeneous turbulence present in a straight square duct. New detailed statistics on particle number concentration and deposition are also obtained and discussed.

The only work available to treat the theory of turbulent flow with suspended particles, this book also includes a section on simulation methods, comparing the model results obtained with the PDF method to those obtained with other techniques, such as DNS, LES and RANS. Written by experienced scientists with background in oil and gas processing, this book is applicable to a wide range of industries -- from the petrol industry and industrial chemistry to food and water processing.

Nanoparticle holds significant promise as the next generation of drug carrier that can realize targeted therapy with minimal toxicity. To improve the delivery efficiency of nanoparticles, it is important to study their transport and deposition in blood flow. Many factors, like particle size, vessel geometry and blood flow rate, have significant influence on the particle transport, thus on the deposition fraction and distribution. In this thesis, computational fluid dynamics (CFD) simulations of blood flow and drug particle deposition were conducted in four models representing the human lung vasculature: artificial artery geometry, artificial vein geometry, original geometry and over-smoothed original geometry. Flow conditions used included both steady-state inlet flow and pulsatile inlet flow. Parabolic flow pattern and lumped mathematic model were used for inlet and outlet boundary conditions respectively. Blood flow was treated as laminar and Newtonian. Particle trajectories were calculated in each of these models by solving the integrated force balance on the particle, and adding a stochastic Brownian term at each step. A receptor-ligand model was integrated to simulate the particle binding probability. The results indicate the following: (i) Pulsatile flow can accelerate the particle binding activity and improve the particle deposition fraction on bifurcation areas; (ii) Unlike drug delivery in lung respiratory system, particle diffusion is very weak in blood flow, no clear relationship between the particle size and deposition area was found in our four-generation lung vascular model; and (iii) Surface
imperfections have the dominant effect on particle deposition fraction over a wide range of particle sizes. Ideal artificial geometry is not sufficient to predict drug deposition, and an accurate image based geometry is required.

The sixth ERCOFTAC Workshop on ‘Direct and Large-Eddy Simulation’ (DLES-6) was held at the University of Poitiers from September 12-14, 2005. Following the tradition of previous workshops in the DLES-series, this edition has reflected the state-of-the-art of numerical simulation of transitional and turbulent flows and provided an active forum for discussion of recent developments in simulation techniques and understanding of flow physics.

This thesis focuses on the numerical investigation of the particle-fluid systems based on the Discrete Element Method (DEM). The whole thesis consists of three parts, in each part we have coupled DEM with different schemes/solvers on the fluid phase. In the first part, we have coupled DEM with Direct Numerical Simulation (DNS) to study the particle-laden turbulent flow. The effect of collisions on the particle behavior in fully developed turbulent flow in a straight square duct was numerically investigated. Three sizes of particles were considered with diameters equal to 50 μm, 100 μm and 500 μm. Firstly, the particle transportation by turbulent flow was studied in the absence of the gravitational effect. Then, the particle deposition was studied under the effect of the free-normal gravity force in which the influence of collisions on the particle resuspension rate and the final stage of particle distribution on the duct floor were discussed, respectively. In the second part, we have coupled DEM with Lattice Boltzmann Method (LBM) to study the particle sedimentation in Newtonian laminar flow. A novel combined LBM-IBM-DEM scheme was presented with its application to model the sedimentation of two-dimensional circular particles in incompressible Newtonian flows. Case studies of single sphere settling in a cavity, and two particles settling in a channel were conducted, where the velocity characteristics of the particles during settling and near the bottom were examined. At last, a numerical example of sedimentation involving 504 particles was finally presented to demonstrate the capability of the combined scheme. Furthermore, a Particulate Immersed Boundary Method (PIBM) for simulating the fluid-particle multiphase flow was presented and assessed in both two and three-dimensional applications. Compared with the conventional IBM, dozens of times speedup in two-dimensional simulation and hundreds of times in three-dimensional simulation can be expected under the same particle and mesh number. Numerical simulations of particle sedimentation in the Newtonian flows were conducted on a combined LBM-PIBM-DEM showing that the PIBM could capture the feature of the particulate flows in fluid and was indeed a promising scheme for the solution of the fluid-particle interaction problems. In the last part, we have coupled DEM with averaged Navier-Stokes equations (NS) to study the particle transportation and wear process on the pipe wall. A case of pneumatic conveying was utilized to demonstrate the capability of the coupling model. The concrete pumping process was then simulated, where the hydraulic pressure and velocity distribution of the fluid phase were obtained. The frequency of the particles impacting on the bended pipe was monitored, a new time average collision intensity model based on impact force was proposed to investigate the wear process of the elbow. The location of maximum erosive wear damage in elbow was predicted. Furthermore, the influences of slurry velocity, bend orientation and angle of elbow on the puncture point location were discussed.

Why is wall turbulence self-sustaining? In this book well-regarded researchers not only discuss what they know and believe, but also speculate on ideas that still require numerical or experimental testing and verification. An initial brief history of boundary layer structure research is followed by chapters on experimental information and specific topics within the subject. There are then sections on computational aspects.

This symposium continues a long tradition for IUGG/IUTAM symposia going back to "Fundamental Problems in Turbulence and their Relation to Geophysics" Marseille, 1961. The five topics that were emphasized were: turbulence modeling, statistics of small scales and coherent structures, convective turbulence, stratified turbulence, and historical developments. The objective was to consider the ubiquitous nature of turbulence in a variety of geophysical problems and related flows. Some history of the contributions of NCAR and its alumni were discussed, including those of Jackson R Herring, who has been a central figure at NCAR since 1972. To the original topics we added rotation, which appeared in many places. This includes rotating stratified turbulence, rotating convective turbulence, horizontal rotation that appears in flows over terrain and the role of small scale vorticity in many flows. These complicated flows have recently begun to be simulated by several groups from around the world and this meeting provided them with an excellent forum for exchanging results, plus inter actions with those doing more fundamental work on rotating stratified and convective flows. New work on double diffusive convection was given in two presentations. The history of Large Eddy Simulations was presented and several new approaches to this field were given. This meeting also spawned some interesting interactions between observational side and how to inter pret the observations with modeling and simulations around
the theme of particle dispersion in these flows.

This book deals with the simulation of the incompressible Navier-Stokes equations for laminar and turbulent flows. The book is limited to explaining and employing the finite difference method. It furnishes a large number of source codes which permit to play with the Navier-Stokes equations and to understand the complex physics related to fluid mechanics. Numerical simulations are useful tools to understand the complexity of the flows, which often is difficult to derive from laboratory experiments. This book, then, can be very useful to scholars doing laboratory experiments, since they often do not have extra time to study the large variety of numerical methods; furthermore they cannot spend more time in transferring one of the methods into a computer language. By means of numerical simulations, for example, insights into the vorticity field can be obtained which are difficult to obtain by measurements. This book can be used by graduate as well as undergraduate students while reading books on theoretical fluid mechanics; it teaches how to simulate the dynamics of flow fields on personal computers. This will provide a better way of understanding the theory. Two chapters on Large Eddy Simulations have been included, since this is a methodology that in the near future will allow more universal turbulence models for practical applications. The direct simulation of the Navier-Stokes equations (DNS) is simple by finite-differences, that are satisfactory to reproduce the dynamics of turbulent flows. A large part of the book is devoted to the study of homogeneous and wall turbulent flows. In the second chapter the elementary concept of finite difference is given to solve parabolic and elliptical partial differential equations. In successive chapters the 1D, 2D, and 3D Navier-Stokes equations are solved in Cartesian and cylindrical coordinates. Finally, Large Eddy Simulations are performed to check the importance of the subgrid scale models. Results for turbulent and laminar flows are discussed, with particular emphasis on vortex dynamics. This volume will be of interest to graduate students and researchers wanting to compare experiments and numerical simulations, and to workers in the mechanical and aeronautic industries.

Durch die gezielte Strukturierung von wärmeübertragenden Oberflächen, wie beispielsweise durch Dellen oder Rippen, kann die örtliche Turbulenz und damit die thermische Durchmischung gesteigert werden. Dies kann die Effizienz von Wärmeübertragern oder Bauteilkühlsystemen erheblich erhöhen. Derartige Oberflächenstrukturierungen begünstigen jedoch das Partikelfouling, daher die Ablagerung suspendierter Partikel, wie z.B. Sand, Schlamm oder Korrosionsprodukte. Gegenstand dieser Arbeit ist die Entwicklung eines universellen, numerischen CFD-Verfahrens zur Vorhersage des partikulären Foulings auf strukturierten Oberflächen, speziell Dellenoberflächen. Das entwickelte Verfahren basiert auf einer Kombination des Lagrangian-Particle-Trackings zur Beschreibung der dispersen Phase (Foulingpartikel), sowie räumlich und zeitlich aufgelöster Large-Eddy Simulation für die Berechnung der kontinuierlichen Phasen (Trägerfluid). Dieses Vorgehen ermöglicht nicht nur die Auswertung der infolge der Partikelablagernungen verminderten thermo-hydraulischen Effizienz, sondern auch die Untersuchung der Wechselwirkungen zwischen turbulenten Strömungsstrukturen und dem partikulären Fouling. Dadurch kann gezeigt werden, dass die Verwendung von sphärischen Dellen als Oberflächenstrukturen nicht nur aus thermo-hydraulischer Sicht die optimale Wahl darstellt, sondern auch eine substantielle Verminderung des Partikelfoulings begünstigt. Die Anwendung von strukturierten heat transfer surfaces, such as dimples or ribs, increases the local turbulence and thus thermal mixing. This can improve the efficiency of heat exchangers or cooling systems significantly. However, structured surfaces are known to promote particulate fouling, hence the unwanted accumulation and deposition of suspended particles (e.g., silt, sludge or iron oxide). The scope of this work is the development of a universal numerical CFD method for the prediction of particulate fouling, especially on dimpled surfaces. The proposed approach is based on a combination of the Lagrangian point-particle tracking for the description of the disperse phase (fouling particles), and spatially and temporally resolved large-eddy simulations for the calculation of the continuous phase (carrier fluid). This approach allows not only the evaluation of the reduced thermo-hydraulic efficiency due to particle deposition, but also the investigation of the interaction between turbulent flow structures and the particulate fouling. It can be shown that the usage of spherical dimples as surface structures is not only the optimal choice from a thermo-hydraulic point of view, but also favors a substantial reduction of particulate fouling.

Further progress in achieving the objectives of the project was made in the period of January I to March 31, 1998. The direct numerical simulation of particle removal process in turbulent gas flows was completed. Variations of particle trajectories are studied. It is shown that the near wall vortices profoundly affect the particle removal process in turbulent boundary layer flows. Experimental data for transport and deposition of fibrous particles in the aerosol wind tunnel was obtained. The measured deposition velocity for irregular fibrous particles is compared with the empirical correlation and the available data for glass fibers and discussed. Additional progress on the sublayer model for evaluating the particle deposition and resuspension in turbulent flows was made.
The human nasal cavities, each with an effective length of only 10cm, feature a wide array of basic flow phenomena due to their complex geometries. Dependent on such airflow fields are the transport and deposition of micro- and nano- particles in the human nasal cavities, of interest to engineers, scientists, air-pollution regulators, and healthcare officials. By utilizing advanced CAD and reverse engineering skills, a realistic model of the human nasal cavity was constructed from MRI image data for 3-D computer simulations. Assuming laminar quasi-steady airflow, dilute micro- and nano-particle suspension flows and local deposition efficiencies were analyzed for 7.5

Presents the numerical simulation results on the 3 generation Weibel lung model and a 3-dimensional morphologically accurate human replica lung model made from MRI scans of a hollow cast taken from autopsy. Velocity profiles, secondary flows, wall shear rate as well as particle deposition were studied. Specific carcinogen particles examined were NNK and BaP, present in cigarette smoke.

This book contains fifty-eight revised and extended research articles written by prominent researchers participating in the Advances in Engineering Technologies and Physical Science conference, held in London, U.K., 4-6 July, 2012. Topics covered include Applied and Engineering Mathematics, Computational Statistics, Mechanical Engineering, Bioengineering, Internet Engineering, Wireless Networks, Knowledge Engineering, Computational Intelligence, High Performance Computing, Manufacturing Engineering, and industrial applications. The book offers the state of art of tremendous advances in engineering technologies and physical science and applications, and also serves as an excellent reference work for researchers and graduate students working on engineering technologies and physical science and applications.

This is an up-to-date review of recent advances in the study of two-phase flows, with focus on gas-liquid flows, liquid-liquid flows, and particle transport in turbulent flows. The book is divided into several chapters, which after introducing basic concepts lead the reader through a more complex treatment of the subjects. The reader will find an extensive review of both the older and the more recent literature, with abundance of formulas, correlations, graphs and tables. A comprehensive (though non exhaustive) list of bibliographic references is provided at the end of each chapter. The volume is especially indicated for researchers who would like to carry out experimental, theoretical or computational work on two-phase flows, as well as for professionals who wish to learn more about this topic.

Understanding the dispersion and the deposition of inertial particles convected by turbulent flows is a domain of research of considerable industrial interest. Inertial particle transport and dispersion are encountered in a wide range of flow configurations, whether they are of industrial or environmental character. Conventional models for turbulent dispersed flows do not appear capable of meeting the growing needs of chemical, mechanical and petroleum industries in this regard and physical environment testing is prohibitive. Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) ha.

Fluid flow and heat transfer processes play an important role in many areas of science and engineering, from the planetary scale (e.g., influencing weather and climate) to the microscopic scales of enhancing heat transfer by the use of nanofluids; understood in the broadest possible sense, they also underpin the performance of many energy systems. This topical Special Issue of Energies is dedicated to the recent advances in the very broad field. This book will be of interest to readers not only in the fields of mechanical, aerospace, chemical, process and petroleum, energy, earth, civil ,and flow instrumentation engineering but, equally, biological and medical sciences, as well as physics and mathematics; that is, anywhere that “fluid flow and heat transfer” phenomena may play an important role or be a subject of worthy research pursuits.

The book provides a broad overview of the full spectrum of state-of-the-art computational activities in multiphase flow as presented by top practitioners in the field. It starts with well-established approaches and builds up to newer methods. These methods are illustrated with applications to a broad spectrum of problems involving particle dispersion and deposition, turbulence modulation, environmental flows, fluidized beds, bubbly flows, and many others.

This technical report discusses the application of the Lattice Boltzmann Method (LBM) and Cellular Automata (CA) simulation in fluid flow and particle deposition. The current work focuses on incompressible flow simulation passing cylinders, in which we incorporate the LBM D2Q9 and CA techniques to simulate the fluid flow and particle loading respectively. For the LBM part, the theories of boundary conditions are studied and verified using the Poiseuille flow test. For the CA part, several models regarding simulation of particles are explained. And a new Digital Differential Analyzer (DDA) algorithm is introduced to simulate particle motion in the Boolean model. The numerical results are compared with a previous probability velocity model by Masselot [Masselot
2000], which shows a satisfactory result.

Presenting tools for understanding the behaviour of gas-liquid flows based on the ways large scale behaviour relates to small scale interactions, this text is ideal for engineers seeking to enhance the safety and efficiency of natural gas pipelines, water-cooled nuclear reactors, absorbers, distillation columns and gas lift pumps. The review of advanced concepts in fluid mechanics enables both graduate students and practising engineers to tackle the scientific literature and engage in advanced research. It focuses on gas-liquid flow in pipes as a simple system with meaningful experimental data. This unified theory develops design equations for predicting drop size, frictional pressure losses and slug frequency, which can be used to determine flow regimes, the effects of pipe diameter, liquid viscosity and gas density. It describes the effect of wavy boundaries and temporal oscillations on turbulent flows, and explains transition between phases, which is key to understanding the behaviour of gas-liquid flows.

This book contains papers presented at the Engineering Foundation Conference on mineral matter in fuels held on November 2-7, 1997 in Kona, Hawaii. The conference is one of a continuing series that was initiated by the CEGB Marwood Engineering Laboratories in 1963. The conference was to be eventually organised by the Engineering Foundation as the need for multi-disciplinary work related to controlling ash effects in combustors became apparent. The conference covers both the science and the applications. The papers also present case histories, particularly for current fuel technologies, developments in advanced technologies for power generation and mathematical modelling of these processes. Developments since 1963 have been slow, but steady, due to the complexity of the chemical and physical processes involved. However, the research presented here displays great improvement in our understanding of the mechanisms by which mineral matter will influence fuel use. Steve Benson from EERC presented a review and current status of issues related to ash deposition in coal combustion and gasification. The application of new analytical tools, which have been detailed in the previous conferences, is presented. These include CCSEM, as well as new techniques for characterising sintering of ash, such as TMA, image analysis, X-ray diffraction crystallography and thermal analysis. The new analytical techniques were extended to encompass widely differing fuels such as biomass. Ole H Larsen from ELSAM Denmark presented a review of these advanced techniques.

Temam and Miranvile present core topics within the general themes of fluid and solid mechanics. The brisk style allows the text to cover a wide range of topics including viscous flow, magnetohydrodynamics, atmospheric flows, shock equations, turbulence, nonlinear solid mechanics, solitons, and the nonlinear Schrödinger equation. This second edition will be a unique resource for those studying continuum mechanics at the advanced undergraduate and beginning graduate level whether in engineering, mathematics, physics or the applied sciences. Exercises and hints for solutions have been added to the majority of chapters, and the final part on solid mechanics has been substantially expanded. These additions have now made it appropriate for use as a textbook, but it also remains an ideal reference book for students and anyone interested in continuum mechanics.

Presented in ten edited chapters this book encompasses important emerging topics in heat transfer equipment, particularly heat exchangers. The chapters have all been selected by invitation only. Advances in high temperature equipment and small scale devices continue to be important as the involved heat transfer and related phenomena are often complex in nature and different mechanisms like heat conduction, convection, turbulence, thermal radiation and phase change as well as chemical reactions may occur simultaneously. The book treats various operating problems, like fouling, and highlights applications in heat exchangers and gas turbine cooling. In engineering design and development, reliable and accurate computational methods are required to replace or complement expensive and time consuming experimental trial and error work. Tremendous advancements in knowledge and competence have been achieved during recent years due to improved computational solution methods for non-linear partial differential equations, turbulence modelling advancement and developments of computers and computing algorithms to achieve efficient and rapid simulations. The chapters of the book thoroughly present such advancement in a variety of applications.

This study is concerned with the modelling of single-phase and two-phase turbulent flows in a square duct over a range of Reynolds numbers with attention focused on the deposition, dispersion and re-suspension of particles. Reynolds averaged Navier Stokes (RANS) modelling is used in conjunction with a Lagrangian particle tracker (LPT). Modelling and simulation of single- and two-phase turbulent flows in circular pipes with the presence of stationary flat beds are also considered using the previously stated methodologies, as well as large eddy simulation (LES). The performance of the RANS modelling technique is evaluated against available experimental, simulation and empirical data. The RANS modelling technique is seen to perform with qualitative
accuracy across all of the test cases considered within this thesis, and it can be said that this approach is capable of reproducing many of the key features associated with these flows. In almost all cases, qualitative agreement is seen between the RANS modelling results and the available experimental data, simulation results and empirical correlations. A key failing of the RANS modelling technique is the inaccurate representation of the magnitude of the secondary velocities found in square ducts and pipes with variable bed height. The RANS modelling technique with a Reynolds stress model (RSM) for turbulence, coupled with a LPT, can be usefully used in modelling particle-laden duct and pipe flow across a range of conditions. Important qualitative information can be gained from this technique in terms of particle deposition, dispersion and re-suspension. For more detailed studies on the physics of these flows, the preferred methodologies are the more advanced simulation techniques of LES and direct numerical simulation (DNS), while there is also a clear need for further experimental investigations of such particle-laden flows.

The more than 90 refereed papers in this volume continue a series of biannual benchmarks for technologies that maximize energy conversion while minimizing undesirable emissions. Covering the entire range of industrial and transport combustion as well as strategies for energy research and development, these state-of-the-art will be indispensable to mechanical and chemical engineers in academia and industry and technical personnel in military, energy and environmental government agencies. The topics covered in this book include wood, oil, gas and coal combustion, combustion of alternative fuels, co-combustion and co-gasification, catalytic combustion, NO, SO, soot fundamentals, advanced diagnostics, burners, fluidized bed combustion, incineration, engines, advanced cycles, gas clean-up, control strategy and clean combustion in process industries.

Copyright code: e4b113f90992dd23f391bae23a56734c