Numerical Simulation Of Low Pressure Die Casting Aluminum

This book will interest researchers, scientists, engineers and graduate students in many disciplines, who make use of mathematical modeling and computer simulation. Although it represents only a small sample of the research activity on numerical simulations, the book will certainly serve as a valuable tool for researchers interested in getting involved in this multidisciplinary field. It will be useful to encourage further experimental and theoretical researches in the above mentioned areas of numerical simulation.

This 2-volumes set contains selected and peer-review papers in the subject areas of engineering thermo physics, applied thermal engineering, power machinery and engineering, fluid engineering and machinery, HVAC, air conditioning and refrigeration, power system and automation, high voltage and insulation technology, motor and electrical, electrical engineering principles and applications, power electronics and power drives, smart grid technologies, power system management.

Computational Fluid Dynamics (CFD) numerical simulations of low-density shock-wave interactions for an incident shock impinging on a cylinder have been performed. Flow-field density gradient and surface pressure and heating define the type of interference pattern and corresponding perturbations. The maximum pressure and heat transfer level and location for various interaction types (i.e., shock-wave incidence with respect to the cylinder) are presented. A time-accurate solution of the Type IV interference is employed to demonstrate the establishment and the steadiness of the low-density flow interaction. Glass, Christopher E. Langley Research Center NASA/TM-1999-209358, NAS 1.15:209358, L-17878

Numerical Simulation

Numerical Simulations

Radio-Frequency Capacitive Discharges

Examples and Applications in Computational Fluid Dynamics

Fluid Dynamics

Pressure fluctuations associated with turbulent boundary layer have been a prominent issue over the past few decades. In order to simulate pressure fluctuations beneath a turbulent boundary layer, a numerical investigation was performed in the current study. Four different turbulence models were employed to calculate the pressure and velocity fluctuations. A new approach of direct numerical simulation (DNS) was developed, as well. The proposed DNS scheme was hybrid of sixth-order weighted compact scheme (WCS) and modified weighted essentially non-oscillatory (WENO) scheme, which is called modified WENO-WCS scheme (MWWS) hereafter. A variety of benchmark problems were investigated to evaluate the accuracy of the proposed numerical scheme. Several empirical/semi-empirical mean square pressure models and single-point wall-pressure spectrum models were investigated to compare mean square wall pressure values. Reynolds-averaged Navier-Stokes based on Spalart-Allmaras (RANS-SA) and Delayed detached-eddy simulation based on Spalart-Allmaras (DDES-SA) turbulence models showed agreement with the Lowson, Lilley and Hodgson, and Goody models. Shear stress transport (RANS-SST) and DDES-SST models showed agreement with the Lowson, Farabee and Casarella, Lilley and Hodgson, and Goody models. The MWWS scheme was in agreement with Lowson and Goody models. Five single-point wall-pressure spectrum models were investigated and compared with numerical results. In low frequency region, results obtained by DDES-SA model and MWWS scheme were in agreement with the Goody model, while RANS-SA, RANS-SST, and DDES-SST turbulence models showed agreement with the Robertson model. In High frequency region, all investigated numerical methods were in agreement with the Goody and Efimtsov (1) models.

In the aviation field there is great interest in high-speed vehicle design. Hypersonic vehicles represent the next frontier of passenger transportation to and from space. However, several design issues must be addressed, including vehicle aerodynamics and aerothermodynamics, aeroshape design optimization, aerodynamic heating, boundary layer transition, and so on. This book contains valuable contributions focusing on hypervelocity aircraft design. Topics covered include hypersonic aircraft aerodynamic and aerothermodynamic design, especially aeroshape design optimization, computational fluid dynamics, and scramjet propulsion. The book also discusses high-speed flow issues and the challenges to achieving the dream of affordable hypersonic travel. It is hoped that the information contained herein will allow for the development of safe and efficient hypersonic vehicles.

CFD numerical simulations of low-density shock-wave interactions for an incident shock impinging on a cylinder have been performed. Flow-field density gradient and surface pressure and heating define the type of interference pattern and corresponding perturbations. The maximum pressure and heat transfer level and location of various interaction types are presented. A time-accurate solution of the Type IV interference is employed to demonstrate the establishment and the steadiness of the low-density flow interaction.

Numerical Simulation for Next Generation Thermal Power Plants

Preconditioning for Numerical Simulation of Low Mach Number Three-dimensional Viscous Turbomachinery Flows

Numerical Simulations of Physical and Engineering Processes
Numerical Simulation of the Aerodynamics of High-Lift Configurations
Numerical Simulation of Convective Fuel Droplet Vaporization and Combustion in a Low Pressure Zero-gravity Environment

This book addresses nearly all aspects of the state of the art in LES & DNS of turbulent flows, ranging from flows in biological systems and the environment to external aerodynamics, domestic and centralized energy production, combustion, propulsion as well as applications of industrial interest. Following the advances in increased computational power and efficiency, several contributions are devoted to LES & DNS of challenging applications, mainly in the area of turbomachinery, including flame modeling, combustion processes and aeroacoustics. The book includes work presented at the tenth Workshop on 'Direct and Large-Eddy Simulation' (DLES-10), which was hosted in Cyprus by the University of Cyprus, from May 27 to 29, 2015. The goal of the workshop was to establish a state of the art in DNS, LES and related techniques for the computation and modeling of turbulent and transitional flows. The book is of interest to scientists and engineers, both in the early stages of their career and at a more senior level.

Pressure wave supercharger is an application of wave rotor technology that utilizes compression waves produced by high-pressure engine exhaust gas to compress the fresh intake air within the channels. The phenomena within the wave rotor channels are governed by compression and expansion waves initiated when the channel ends are periodically exposed to differing pressure ports. Two incoming fluids are brought into contact for a very short amount of time to facilitate efficient energy and momentum transfer, thereby exchanging pressure dynamically between the fluids by means of unsteady pressure waves. Since the energy transfer is based on unsteady pressure waves, correct matching of waves and ports is essential for optimum results. Mistiming of the waves in the channels is detrimental to the efficient exchange of pressure and low-pressure exhaust scavenging, which ensures minimum exhaust gas recirculation. Due to varying speed and load conditions of the unit to be supercharged, it is not always possible to maintain the rotor speed constant at the design point. To mitigate the effects of wave mistiming due to varying speed, a well-designed combination of wall-pockets was used in Comprex® pressure wave supercharger. The wall-pockets are the recesses provided in the endplates of pressure wave superchargers to create necessary pressure zones at desired locations. This thesis details an extensive qualitative and computational investigation of the performance of pressure wave superchargers with pockets. Numerical simulations of pressure wave superchargers have been performed using the wave rotor analysis codes employed at the Combustion and Propulsion Research Laboratory at IUPUI. This work also pays close attention to inspecting the numerical schemes and modeling of different physical phenomena used in each code. A comparative verification of the wave rotor analysis codes has been conducted to ensure that the same fundamental numerical scheme is correctly implemented in each code. The issue of low-pressure scavenging has been demonstrated by simulating the four-port (pocketless) pressure wave supercharger operating at lower speeds. The wall-pockets have been modeled using a simple lumped volume technique. The gas state in the lumped volume of pockets is estimated using the continuity and energy equations such that the net mass and energy fluxes between each pocket and the wave rotor channels are close to zero. The lumped volume models of pockets have been implemented in the four-port wave rotor configurations to simulate the pressure wave superchargers with pockets. The simulation results show that the pockets assist to maintain sufficient pressure in the desired zones to facilitate proper low-pressure scavenging during lower rotor speed operations. The Comprex simulation results have been observed to be in good agreement with experimental data and qualitative analysis. Specific observations on the performance of each code and comprehensive simulation results have been presented.

The book provides highly specialized researchers and practitioners with a major contribution to mathematical models’ developments for energy systems. First, dynamic process simulation models based on mixture flow and two-fluid models are developed for combined-cycle power plants, pulverised coal-fired power plants, concentrated solar power plant and municipal waste incineration. Operation data, obtained from different power stations, are used to investigate the capability of dynamic models to predict the behaviour of real processes and to analyse the influence of modeling assumptions on simulation results. Then, a computational fluid dynamics (CFD) simulation programme, so-called DEMEST, is developed. Here, the fluid-solid, particle-particle and particle-wall interactions are modeled by tracking all individual particles. To this purpose, the deterministic Euler-Lagrange/Discrete Element

Method (DEM) is applied and further improved. An emphasis is given to the determination of inter-phase values, such as volumetric void fraction, momentum and heat transfers, using a new procedure known as the offset-method and to the particle-grid method allowing the refinement of the grid resolution independently from particle size. Model validation is described in detail. Moreover, thermochemical reaction models for solid fuel combustion are developed based on quasi-single-phase, two-fluid and Euler-Lagrange/MP-PIC models. Measurements obtained from actual power plants are used for validation and comparison of the developed numerical models.

Numerical Simulation of Complex Turbomachinery Flows

Numerical Simulation of Low-pressure Explosive Combustion in Compartment Fires

A Combined CFD-DSMC Method for Numerical Simulation of Nozzle Plume Flows

A Framework for the Direct Numerical Simulation of Phase Change Processes of Water at Low Temperature and Pressure

Numerical Simulation of a Low Emissions Gas Turbine Combustor Using KIVA-2

A numerical simulation has been constructed to obtain a detailed, quantitative estimate of the electromagnetic fields generated in a recently-proposed collective accelerator scheme for electrons. The code treats the secondary electrons by particle simulation and the beam dynamics by a time-dependent envelope model. The simulation gives a fully relativistic description of secondary electrons moving in selfconsistent electromagnetic fields. The calculations are made using coordinates t, x, y, z for the electrons and t, ct-z, r for the axisymmetric electromagnetic fields and currents. Code results showing the axial electric field dependence on the configuration of the ultrashort U.V. laser pulse will be given. 4 refs., 4 figs.

This thesis focuses on the numerical simulation of Low-Tension-Gas (LTG) process, that involves the injection of surfactant and gas to generate and propagate foam for mobility control, and to mobilize the residual oil to waterflood by reducing the interfacial tension between oil and water. This EOR process, is an alternative to surfactant-polymer process and is applicable to challenging conditions including tight formations, high temperature and high salinity reservoirs where polymer implementation is not feasible due to physical and/or economic constraints. In this study, the experimental data used for numerical simulation involve tight carbonate rock with high formation salinity. For the numerical simulation study, a LTG model developed by the Uni- versity of Texas at Austin and incorporated into the compositional equation- of-state CMG/GEM simulator is utilized. The model includes the modeling of IFT reduction, surfactant partitioning, relative permeability, foam, and ad- sorption. In some case, a numerical simulation model may involve a large number of uncertain parameters, which often exceeds the experimental data available. Hence, there may exist more than one combination of the parameters that provide a good agreement between the model and the experiments. Therefore, a numerical simulation study is undertaken in order to develop a methodology for determining the LTG model parameters through a series of simulations and data-fitting of strategically selected experimental data to reduce the non- uniqueness of the problem while preserving the physics of the process. Low capillary number water-oil relative permeability parameters are determined through matching of waterflood experimental data, which is a pre- liminary procedure of LTG flooding. In addition, the reference foam mobility reduction factor, the dry-out function, and the gas relative permeability curve are estimated through matching of foam quality tests. Thereafter, a sensitivity analysis of the remaining uncertain parameters is performed to investigate the significance of the parameters on the oil recovery and pressure drop. Data- fitting of the LTG flood experimental data is then performed to determine es- timations for the rest of the parameter space, including surfactant adsorption, dispersivity, intermediate capillary number and associated oil-water relative permeability curves, oil-gas curvature, and the rest of the foam parameters. In conclusion, this thesis provides a methodology for estimating rela- tive permeability, foam strength, adsorption and dispersivity parameters for LTG simulation. These findings will be proven useful for understanding LTG flooding behavior in EOR processes.

Computational Techniques for Multiphase Flows, Second Edition, provides the latest research and theories covering the most popular multiphase flows The book begins with an overview of the state-of-the-art techniques for multiple numerical methods in handling multiphase flow, compares them, and finally highlights their strengths and weaknesses. In addition, it covers more straightforward, conventional theories and governing equations in early chapters, moving on to the more modern and complex computational models and tools later in the book. It is therefore accessible to those who may be new to the subject while also featuring topics of interest to the more experienced researcher. Mixed or multiphase flows of solid/liquid or solid/gas are commonly found in many industrial fields, and their behavior is complex and difficult to predict in many cases. The use of computational fluid dynamics (CFD) has emerged as a powerful tool for understanding fluid mechanics in multiphase reactors, which are widely used in the chemical, petroleum, mining, food, automotive, energy, aerospace and pharmaceutical industries. This revised edition is an ideal reference for scientists, MSc students and chemical and mechanical engineers in these areas. Includes updated chapters in addition to a brand-new section on granular flows. Features novel solution methods for multiphase flow, along with recent case studies. Explains how and when to use the featured technique and how to interpret the results and apply them to improving applications.

Direct and Large-Eddy Simulation X

Advances in Power and Electrical Engineering

Theory, Computation, and Numerical Simulation

Numerical Simulation of Shock Induced Combustion

Numerical Simulation, An Art of Prediction, Volume 2

Ready access to computers at an institutional and personal level has defined a new era in teaching and learning. The opportunity to extend the subject matter of traditional science and engineering disciplines into the realm of scientific computing has become not only desirable, but also necessary. Thanks to port ability and low overhead and operating costs, experimentation by numerical simulation has become a viable substitute, and occasionally the only alternative, to physical experiment at ion. The new environment has motivated the writing of texts and mono graphs with a modern perspective that incorporates numerical and com puter programming aspects as an integral part of the curriculum: meth ods, concepts, and ideas should be presented in a unified fashion that motivates and underlines the urgency of the new elements, but does not compromise the rigor of the classical approach and does not oversimplify. Interfacing fundamental concepts and practical methods of scientific computing can be done on different levels. In one approach, theory and implement at ion are kept complementary and presented in a sequential fashion. In a second approach, the coupling involves deriving compu tational methods and simulation algorithms, and translating equations into computer code instructions immediately following problem formu lations. The author of this book is a proponent of the second approach and advocates its adoption as a means of enhancing learning: interject ing methods of scientific computing into the traditional discourse offers a powerful venue for developing analytical skills and obtaining physical insight.

Numerical Simulation of Low-pressure Explosive Combustion in Compartment FiresNumerical Simulations of Physical and Engineering ProcessesIntechOpen

This book deals with numerical simulations and computations of the turbulent flow around high-lift configurations commonly used in aircraft. It is devoted to the Computational Fluids Dynamics (CFD) method using full Navier-Stokes solvers typically used in the simulation of high-lift configuration. With the increase of computational resources in the aeronautical industry, the computation of complex flows such as the aerodynamics of high-lift configurations has become an active field not only in academic but also in industrial environments. The scope of the book includes applications and topics of interest related to the simulation of high-lift configurations such as: lift and drag prediction, unsteady aerodynamics, low Reynolds effects, high performance computing, turbulence modelling, flow feature visualization, among others. This book gives a description of the state-of-the-art of computational models for simulation of high-lift configurations. It also shows and discusses numerical results and validation of these computational models. Finally, this book is a good reference for graduate students and researchers interested in the field of simulation of high-lift configurations.

Numerical Simulation of Secondary Electron Orbits Near an Electron Beam Propagating in a Low Pressure Gas

TILDA: Towards Industrial LES/DNS in Aeronautics

Nuclear Science Abstracts

Applications to Electric Double Layers

Numerical Modeling of Low-pressure Plasmas

This book systematically introduces readers to the simulation theory and techniques of multiple media for unconventional tight reservoirs. It summarizes the macro/microscopic heterogeneities; the features of multiscale multiple media; the characteristics of complex fluid properties; the occurrence state of continental tight oil and gas reservoirs in China; and the complex flow characteristics and coupled production mechanism under unconventional development patterns. It also discusses the simulation theory of multiple media for unconventional tight oil and gas reservoirs; mathematic model of flow through discontinuous multiple media; geological modeling of discrete multiscale multiple media; and the simulation of multiscale, multiphase flow regimes and multiple media. In addition to the practical application of simulation and software for unconventional tight oil and gas, it also explores the development trends and prospects of simulation technology. The book is of interest to scientific researchers and technicians engaged in the development of oil and gas reservoirs, and serves as a reference resource for advanced graduate students in fields related to petroleum.

Nowadays mathematical modeling and numerical simulations play an important role in life and natural science. Numerous researchers are working in developing different methods and techniques to help understand the behavior of very complex systems, from the brain activity with real importance in medicine to the turbulent flows with important applications in physics and engineering. This book presents an overview of some models, methods, and numerical computations that are useful for the applied research scientists and mathematicians, fluid tech engineers, and postgraduate students.

Presenting papers from the 2013 annual meeting of The Minerals, Metals & Materials Society (TMS), this volume covers developments in all aspects of high temperature electrochemistry, from the fundamental to the empirical and from the theoretical to the applied.

TMS 2013 142nd Annual Meeting and Exhibition, Annual Meeting

Numerical Simulation of Wall-pressure Fluctuations Due to Turbulent Boundary Layer
Numerical Simulation of Low-density Shock-wave Interactions
Unconventional Tight Reservoir Simulation: Theory, Technology and Practice
Numerical Simulation of Pressure Wave Supercharger with Pockets Operating at Different Speeds

Small rockets are used extensively for attitude control and other low thrust operations on trans-atmospheric vehicles and spacecraft. Low thrust electric thrusters and a variety of chemical thrusters are being developed for these applications. Several experimental and numerical studies have been directed towards the prediction and characterization of the fluid dynamics and kinetics of the nozzle and plume flows.

Numerical simulation is a technique of major importance in various technical and scientific fields. Whilst engineering curricula now include training courses dedicated to it, numerical simulation is still not well-known in some economic sectors, and even less so among the general public. Simulation involves the mathematical modeling of the real world, coupled with the computing power offered by modern technology. Designed to perform virtual experiments, digital simulation can be considered as an "art of prediction". Embellished with a rich iconography and based on the testimony of researchers and engineers, this book shines a light on this little-known art. It is the second of two volumes and gives examples of the uses of numerical simulation in various scientific and technical fields: agriculture, industry, Earth and universe sciences, meteorology and climate studies, energy, biomechanics and human and social sciences.

Numerical simulations of particle image velocimetry (PIV) experiments conducted with vortex generating jets (VGJs) on a flat plate, at a Reynolds number based on plate length of 50,000, were performed for three flow conditions using a time-accurate hybrid Navier-Stokes solver. Time-averaged steady blowing of angled jets, subjected to a zero pressure gradient, yielded excellent agreement with the PIV data in terms of vortex formation and strength. Observed flow features include primary and secondary vortices, where the primary vortex eventually dominates the downstream region. A shell wall structure, created by smaller vortical structures surrounding the developing vortices, was also observed. A pulsed jet in a zero pressure gradient was then initialized from a no-control case. A qualitative comparison between averaged experimental and instantaneous numerical results was performed with good agreement in terms of the convected size and distance of the wake. Analysis of the instantaneous numerical flow field agreed well with various flow visualization experiments describing the formation of "kidney" vortices. Various indicators point to the production of a primary vortex by the reduced mass flow of the pulsed jet. Finally, an adverse pressure gradient was applied, inducing a laminar separation zone on the plate. A pulsed angled jet induced strong spanwise vortices in the separated shear layer which appear to weaken the separation zone and allow the bulk jet fluid to flush the remaining low-momentum fluid out of the domain. It is reasonable to assume that reduced blowing ratios and duty cycles would produce similar shear layer vortices and comparable loss reductions. Influences of both turbulent transition and dominant vortical structures were observed, though the spanwise shear layer vortices appear to be critical to the laminar separation reduction scenarios observed in this study.

Geohydrology and Numerical Simulation of the Ground-water Flow System of Kona, Island of Hawaii
Numerical Simulation of Tropical Cyclone Development with Latent Heat Release by the Resolvable Scales
Numerical Simulation of Cellular Blood Flow
From Brain Imaging to Turbulent Flows
A Numerical Simulation of Hurricane Landfall

High Hydrostatic Pressure Processing (HHPP) is a novel non-thermal food processing technology for producing safe, high quality food products, with minimum detrimental effects of thermal processing such as loss of original flavor and color. The high pressure range used for processing food products is 100 to 1000 MPa. Clams are high pressure processed in the range of 200-350 MPa and fruit juices between 300-600 MPa. Spore found mainly in low acid foods, and prions need even higher pressures for inactivation. When pressure is applied on a food product using liquid medium, adiabatic heat generation occurs due to compression of the pressurizing medium and the food product, which results in increase in their temperatures. This increase in temperature is different for different foods. For example, water heats up by 2-3°C per 100 MPa increase in pressure. Oils and fats heat more (6-9°C) due to their higher compressibility, lower thermal conductivity, and lower heat capacity. In a high pressure process, the heat generated by adiabatic compression is continuous dissipated to the thick metal wall of the vessel during pressurization and pressure hold stages. The heat loss at the wall and the natural convection flow near the vessel wall give rise to non-uniform temperature distribu within the pressurization medium. Therefore, the objective of this research was to carry out numerical simulation of thermal transport in pressurizing medium (water) during HHPP (at room temperature and higher initia temperature) to predict the temperature distribution. Numerical predictions were validated using experimental data. The impact of the response time of the high pressure thermocouple assembly on the measured trans temperature response was taken into account. Results obtained from the numerical simulation showed that the temperature distribution in the pressurizing medium became non-uniform during the high pressure proces and this non-uniformity increased with increasing initial temperatures. Also, increasing the vessel size and inserting an insulating sleeve in the vessel decreased the non-uniformity in temperature. Non-uniformity in temperature in the pressurizing medium can lead to non-uniform microbial inactivation and is of most relevance when a combination of high pressure and high temperature is used to inactivate spores.

Vanadium pentoxide (V2O5) was chosen as a sintering aid to lower the sintering temperature of the ZnO–TiO2 system. The effect of V2O5 on the sintering behavior of ZnO–TiO2 ceramics and cermets was studied as function of additive percentage and sintering temperature. Then porous alumina ceramics with different porosity were fabricated by two different methods. For both of two methods, the bulk density, porosity, and microstructure of the obtained alumina ceramics were studied. Open cell foams, as one type of porous materials, are nowadays commercially used in a broad range of applications. A method for the generation of rand foam structures, based on Laguerre-Voronoi tessellations of randomly packed spheres with log-normal volume distribution, was proposed. Then a three dimensional random Laguerre-Voronoi foam model was develop investigate the elastic properties of open cell ceramic foams. Finally, the pressure drop and heat transfer through open cell foams were investigated. A pressure drop correlation with a universal form was developed ba theoretical grounds and the tortuosity of open cell foams was taken into account for the pressure drop. The developed correlation was validated using numerical simulations. Permeability and friction factor of foams we also evaluated. Then the computed interstitial heat transfer coefficients were investigated. Furthermore, a correlation was given to derive the hydraulic diameter from the ppi number.

Numerical Simulations of Physical and Engineering Process is an edited book divided into two parts. Part I devoted to Physical Processes contains 14 chapters, whereas Part II titled Engineering Processes has 13 contributions. The book handles the recent research devoted to numerical simulations of physical and engineering systems. It can be treated as a bridge linking various numerical approaches of two closely inter-related branches of science, i.e. physics and engineering. Since the numerical simulations play a key role in both theoretical and application oriented research, professional reference books are highly needed by pure research scientists, applied mathematicians, engineers as well post-graduate students. In other words, it is expected that the book will serve as an effective tool in training the mentioned groups of researchers and beyond.
Numerical Simulation of Low-Density Shock-Wave Interactions
Low-temperature Sintering and Fabrication Research of Ceramics and Numerical Simulation on Elastic, Pressure Drop and Heat Transfer Properties of Open Cell Foams
Propagating small scale features observed in the prehurricane phase. II.
Examples

The first publication of its kind in the field, this book describes comprehensively and systematically radio-frequency (rf) capacitive gas discharges of intermediate and low pressure and their application to gas laser excitation and to plasma processing. Text presents the physics underlying rf discharges along with techniques for obtaining such discharges, experimental methods and results, and theoretical and numerical modeling findings. Radio-Frequency Capacitive Discharges is written by well-known specialists in the field, authors of many theoretical and experimental works. They provide simple and clear discussions of complicated physical phenomena. A complete review on the state of the art is included. This interesting new book can be used as a textbook for students and postgraduates and as a comprehensive guidebook by specialists.
"Inductive plasmas are simulated by using a one-dimensional particle-in-cell simulation including Monte Carlo collision techniques (pic/mcc). To model inductive heating, a non-uniform radio-frequency (rf) electric field, perpendicular to the electron motion is included into the classical particle-in-cell scheme. The inductive plasma pic simulation is used to confirm recent experimental results that electric double layers can form in current-free plasmas. These results differ from previous experimental or simulation systems where the double layers are driven by a current or by imposed potential differences. The formation of a super-sonic ion beam, resulting from the ions accelerated through the potential drop of the double layer and predicted by the pic simulation is confirmed with nonperturbative laser-induced fluorescence measurements of ion flow. It is shown that at low pressure, where the electron mean free path is of the order of, or greater than the system length, the electron energy distribution function (eedf) is close to Maxwellian, except for its tail which is depleted at energies higher than the plasma potential. Evidence supporting that this depletion is mostly due to the high-energy electrons escaping to the walls is given. A new hybrid simulation scheme (particle ions and Boltzmann/particle electrons), accounting for non-Maxwellian eedf and self-consistently simulating low-pressure high-density plasmas at low computational cost is proposed. Results obtained with the 'improved' hybrid model are in much better agreement with the full pic simulation than the classical non self-consistent hybrid model. This model is used to simulate electronegative plasmas and to provide evidence supporting the fact that propagating double layers may spontaneously form in electronegative plasmas. It is shown that critical parameters of the simulation were very much aligned with critical parameters of the experiment."
In order to simulate cellular blood, a coarse-grained spectrin-link (SL) red blood cell (RBC) membrane model is coupled with a lattice-Boltzmann (LB) based suspension solver. The LB method resolves the hydrodynamics governed by the Navier--Stokes equations while the SL method accurately models the deformation of RBCs under numerous configurations. This method has been parallelized using Message Passing Interface (MPI) protocols for the simulation of dense suspensions of RBCs characteristic of whole blood on world-class computing resources. :Simulations were performed to study rheological effects in unbounded shear using the Lees-Edwards boundary condition with good agreement with rotational viscometer results from literature. The particle-phase normal-stress tensor was analyzed and demonstrated a change in sign of the particle-phase pressure from low to high shear rates due to RBCs transitioning from a compressive state to a tensile state in the flow direction. Non-Newtonian effects such as viscosity shear thinning were observed for shear rates ranging from 14-440 inverse seconds as well as the strong dependence on hematocrit at low shear rates. An increase in membrane bending energy was shown to be an important factor for determining the average orientation of RBCs, which ultimately affects the suspension viscosity. The shear stress on platelets was observed to be higher than the average shear stress in blood, which emphasizes the importance of modeling platelets as finite particles.
Numerical Simulation Study of Low-Tension-Gas (LTG) Flooding for Enhanced Oil Recovery in Tight Formations
Past, Present and Future Developments
Numerical Simulation of Thermal Transport in a High Hydrostatic Pressure Food Processing Vessel
Numerical Simulation of Vortex Generating Jets in Zero and Adverse Pressure Gradients
Paving the Way for Future Accurate CFD - Results of the H2020 Research Project TILDA, Funded by the European Union, 2015 -2018
An unsteady, multiblock, Reynolds Averaged Navier Stokes solver based on Runge-Kutta scheme and Pseudo-time step for turbo-machinery applications was developed. The code was validated and assessed against analytical and experimental data. It was used to study a variety of physical mechanisms of unsteady, three-dimensional, turbulent, transitional, and cooling flows in compressors and turbines. Flow over a cylinder has been used to study effects of numerical aspects on accuracy of prediction of wake decay and transition, and to modify K-epsilon models. The following simulations have been performed: (a) Unsteady flow in a compressor cascade: Three low Reynolds number turbulence models have been assessed and data compared with

Euler/boundary layer predictions. Major flow features associated with wake induced transition were predicted and studied; (b) Nozzle wake-rotor interaction in a turbine: Results compared to LDV data in design and off-design conditions, and cause and effect of unsteady flow in turbine rotors were analyzed; (c) Flow in the low-pressure turbine: Assessed capability of the code to predict transitional, attached and separated flows at a wide range of low Reynolds numbers and inlet freestream turbulence intensity. Several turbulence and transition models have been employed and comparisons made to experiments; (d) leading edge film cooling at compound angle: Comparisons were made with experiments, and the flow physics of the associated vortical structures were studied; and (e) Tip leakage flow in a turbine. The physics of the secondary flow in a rotor was studied and sources of loss identified. Chernobrovkin, A. A. and Lakshiminarayana, B. Glenn Research Center NAG3-1736; NAG3-2025; RTOP 523-26-33...

This book offers detailed insights into new methods for high-fidelity CFD, and their industrially relevant applications in aeronautics. It reports on the H2020 TILDA project, funded by the European Union in 2015-2018. The respective chapters demonstrate the potential of high-order methods for enabling more accurate predictions of non-linear, unsteady flows, ensuring enhanced reliability in CFD predictions. The book highlights industrially relevant findings and representative test cases on the development of high-order methods for unsteady turbulence simulations on unstructured grids; on the development of the LES/DNS methodology by means of multilevel, adaptive, fractal and similar approaches for applications on unstructured grids; and on leveraging existent large-scale HPC networks to facilitate the industrial applications of LES/DNS in daily practice. Furthermore, the book discusses multidisciplinary applications of high-order methods in the area of aero-acoustics. All in all, it offers timely insights into the application and performance of high-order methods for CFD, and an extensive reference guide for researchers, graduate students, and industrial engineers whose work involves CFD and turbulence modeling.

Computational Techniques for Multiphase Flows

Hypersonic Vehicles