Cfd Modelling Of A Horizontal Three Phase Separator A

This paper assesses the accuracy of the simplified frame cavity conduction/convection and radiation models presented in ISO 15099 and used in software for rating and labeling window products. Temperatures and U-factors for typical horizontal window frames with internal cavities are compared; results from Computational Fluid Dynamics (CFD) simulations with detailed radiation modeling are used as a reference. Four different frames were studied. Two were made of polyvinyl chloride (PVC) and two of aluminum. For each frame, six different simulations were performed, two with a CFD code and four with a building-component thermal-simulation tool using the Finite Element Method (FEM). This FEM tool addresses convection using correlations from ISO 15099; it addressed radiation with either correlations from ISO 15099 or with a detailed, view-factor-based radiation model. Calculations were performed using the CFD code with and without fluid flow in the window frame cavities; the calculations without fluid flow were performed to verify that the CFD code and the building-component thermal-simulation tool produced consistent results. With the FEM-code, the practice of subdividing small frame cavities was examined, in some cases not subdividing, in some cases subdividing cavities with interconnections smaller than five millimeters (mm) (ISO 15099) and in some cases subdividing cavities with interconnections smaller than seven mm (a breakpoint that has been suggested in other studies). For the various frames, the calculated U-factors were found to be quite comparable (the maximum difference between the reference CFD simulation and the other simulations was found to be 13.2 percent). A maximum difference of 8.5 percent was found between the CFD simulation and the FEM simulation using ISO 15099 procedures. The ISO 15099 correlation works best for frames with high U-factors. For more efficient frames, the relative differences among various simulations are larger. Temperature was also compared, at selected locations on the frames. Small differences was found in the results from model to model. Finally, the effectiveness of the ISO cavity radiation algorithms was examined by comparing results from these algorithms to detailed radiation calculations (from both programs). Our results suggest that improvements in cavity heat transfer calculations can be obtained by using detailed radiation modeling (i.e. view-factor or ray-tracing models), and that incorporation of these strategies may be more important for improving the accuracy of results than the use of CFD modeling for horizontal cavities.
CFD Modelling of Fluid Hydrodynamics in Horizontal Primary SeparatorThe Study of Two-phase Flow for Crude Oil Production Using Computational Fluid Dynamics (CFD)

Aero engine bearing chambers are complex machine elements inside the engines, supporting up to three concentric shafts on bearings. For safety reasons, the aero engines always employ rolling-element type bearings and therefore require a sufficient oil supply for lubrication in order to guarantee a

reliable operation. As a consequence, a complex two-phase flow consisting of oil and sealing air governs the bearing chambers. A highly dynamic oil film, flowing along the chamber walls, plays a vital role to fulfill the tasks of cooling, lubricating and cleaning the bearing chambers. The design and optimization process of the bearing chambers requires a detailed understanding in order to accurately simulate the film behaviour inside the bearing chambers. Based on the earlier experimental investigations, it is known that near the scavenge off-take a relatively thick film exists. The numerical model to simulate these films must therefore take into account the elliptical behaviour of such films. Among the different models, the Volume Of Fluid (VOF) Model offers the best compromise between accuracy and efforts. However, preliminary attempts to model a fully developed and turbulent test case from literature revealed an unphysical pressure drop and velocity profile in the gas phase above the film flow. An inadequate turbulence modelling near the gas-liquid interface was identified as the problem source. The 2-Equation turbulence models (k- epsilon & k- omega) were extended to achieve a substantial improvement.
This book contains twenty-one original papers and one review paper published by internationally recognized experts in the Atmosphere Special Issue "Recent Advances in Urban Ventilation Assessment and Flow Modelling", years 2017–2019. The Special Issue includes contributions on recent experimental and modelling works, techniques, and developments mainly tailored to the assessment of urban ventilation on flow and pollutant dispersion in cities. The study of ventilation is of critical importance, as it addresses the capacity with which a built urban structure is capable of replacing the polluted air with ambient fresh air. Here, ventilation is recognized as a transport process that improves local microclimate and air quality and closely relates to the term “breathability”. The efficiency with which street canyon ventilation occurs depends on the complex interaction between the atmospheric boundary layer flow and the local urban morphology. The individual contributions to this Issue are summarized and categorized into four broad topics: (1) outdoor ventilation efficiency and application/development of ventilation indices, (2) relationship between indoor and outdoor ventilation, (3) effects of urban morphology and obstacles to ventilation, and (4) ventilation modelling in realistic urban districts. The results and approaches presented and proposed will be of great interest to experimentalists and modelers, and may constitute a starting point for the improvement of numerical simulations of flow and pollutant dispersion in the urban environment, for the development of simulation tools, and for the implementation of mitigation strategies.
This book comprises select proceedings of the International Conference on Future Learning Aspects of Mechanical Engineering (FLAME 2018). The book gives an overview of recent developments in the field of thermal and fluid engineering, and covers theoretical and experimental fluid dynamics, numerical methods in heat transfer and fluid mechanics, different modes of heat transfer,

multiphase transport and phase change, fluid machinery, turbo machinery, and fluid power. The book is primarily intended for researchers and professionals working in the field of fluid dynamics and thermal engineering.

This paper shows the study of a new artificial lift system, known as GALLOP, for horizontal gas well deliquifaction using CFD software. To begin with, the casing is filled with water coming from the reservoir due to changes in the inner pressure causing the valves to open. Afterwards, gas is injected in the double pipe system in order to expel the water accumulated. For the simulations there were three conditions of gas injection considered and the system was initially assumed to be completely full of water. A low, a mid, and high gas injection were included, being 30, 50 and 70 CFM, respectively. Additionally, the valves on the mandrel were simulated both as open or closed. The simulations were conducted using STAR-CCM+ v12.02 software with a VOF model (Volume of Fluid model). Also, due to computational cost and results quality, a normal mesh was implemented. All simulations showed that the gas was able to travel all the way through the system but did not expel the water completely, mainly because of gravitational force and the extended height of the vertical pipe. As it was expected, the highest injection rate was able to reach the vertical outlet first and thus had the most stable void fraction. Likewise, it was clear that the void fraction was more stable when the valves were closed, indicating that the liquid was removed successfully. Additionally, all of the injection rates showed a clear annular pattern in the horizontal piping. On the other hand, the pressure drop for an injection of 30 CFM with closed valve was compared to experimental data showing an error of 34.4 %, where more experimental data must be conducted, a finer mesh, and more specialized mathematical model must be implemented to achieve more accurate results. Lastly, the other two conditions were compared using the non-slip model since there is no experimental data yet available.

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.

Modeling the liquid-gas phase flow inside the horizontal and inclined pipe using CFD analysis is difficult due to continuously changing flow patterns. The main objectives of this research are to investigate the flow pattern of liquid-gas phase inside the horizontal and inclined pipe. Two-phase flows specifically on the liquidgas flow have a complex flow pattern that can be observed by develop the 3- Dimensional model using the Computational Fluid Dynamic (CFD) software that consist of Gambit for develop the model of horizontal and inclined pipe and then transfer the data to Fluent for further analysis. The simulation was conducted by modelling the horizontal and inclined plane with the length of 7 m and 0.08 m of inner diameter. This simulation was carried out under adiabatic condition and operating at normal temperature which was 298 K. The gravity was enabled in order to differentiate the phase flow inside the horizontal and incline pipe due to the density of liquid-gas phases. The simulation was run using the Volume of Fluid (VOF) for the solver. The manipulated variables which were velocity of the liquid and gas are been changed in order to predict the various flow pattern for both horizontal and inclined pipe. The results of flow pattern are been analyzed and compared with the previous researchers' results. This can be concluded that all flow patterns appearing in the Baker chart can be simulated using existing CFD. In order to improve the effectiveness of the model developed, the simulation needed to be run until the iteration is converging.

By far the most commonly encountered and energy-intensive unit operation in almost all industrial sectors, industrial drying continues to attract the interest of scientists, researchers, and engineers. The Handbook of Industrial Drying, Fourth Edition not only delivers a comprehensive treatment of the current state of the art, but also serves as a

Multiphase flows are found in all areas of technology, at all length scales and flow regimes and can involve compressible or incompressible linear or nonlinear, fluids. However, although they are ubiquitous, multiphase flows continue to be one of the most challenging areas of computational mechanics, with numerous problems as yet unsolved. Advanced computational and experimental methods are often required to solve the equations that describe such complex problems. The many challenges that must be faced in solving them include modelling nonlinear fluids, modelling and tracking interfaces, dealing with multiple length scales, characterising phase structures, and treating drop break-up and coalescence. It is important to validate models, which calls for the use of expensive and difficult experimental techniques.This book presents contributions on the latest research in the techniques for solving multiphase flow problems, presented at the seventh in a biennial series of conferences on the subject that began in 2001. Featured topics include: Flow in porous media; Turbulent flow; Multiphase flow simulation; Image processing; Heat transfer; Atomization; Interface behaviour; Oil and gas applications; Experimental measurements; Energy applications; Biological flows; Micro and macro fluids; Compressible flows.

In the wake of energy crisis due to rapid growth of industries, urbanization, transportation, and human habit, the efficient transfer of heat could play a vital role in energy saving. Industries, household requirements, offices, transportation are all dependent on heat exchanging equipment. Considering these, the present book has incorporated different sections related to general aspects of heat transfer phenomena, convective heat transfer mode, boiling and condensation, heat transfer to two phase flow and heat transfer augmentation by different means.

This book gathers outstanding papers on numerical modeling in Mechanical Engineering (Volume 2) as part of the proceedings of the 1st International Conference on Numerical Modeling in Engineering (NME 2018), which was held in Ghent, Belgium. The overall objective of the conference was to bring together international scientists and engineers in academia and industry from fields related to advanced numerical techniques, such as the finite element method (FEM), boundary element method (BEM), isogeometric analysis (IGA), etc., and their applications to a wide range of engineering disciplines. This book addresses various industrial engineering applications of numerical simulations to Mechanical and Materials Engineering, including: Aerospace applications, Acoustic analysis, Biomechanical applications, Contact problems and wear, Heat transfer analysis, Vibration and dynamics, Transient analysis, Nonlinear analysis, Composite materials, Polymers, Metal alloys, Fracture mechanics, Fatigue of materials, Creep behavior, Phase transformation, and Crystal plasticity.

This book contains the successful submissions to a Special Issue of Energies entitled “Engineering Fluid Dynamics 2019–2020”. The topic of engineering fluid dynamics includes both experimental and computational studies. Of special interest were submissions from the fields of mechanical, chemical, marine, safety, and energy engineering. We welcomed original research articles and review articles. After one-and-a-half years, 59 papers were submitted and 31 were accepted for publication. The average processing time was about 41 days. The authors had the following geographical distribution: China (15); Korea (7); Japan (3); Norway (2); Sweden (2); Vietnam (2); Australia (1); Denmark (1); Germany (1); Mexico (1); Poland (1); Saudi Arabia (1); USA (1); Serbia (1). Papers covered a wide range of topics including analysis of free-surface waves, bridge girders, gear boxes, hills, radiation heat transfer, spillways, turbulent flames, pipe flow, open channels, jets, combustion chambers, welding, sprinkler, slug flow, turbines, thermoelectric power generation, airfoils, bed formation, fires in tunnels, shell-and-tube heat exchangers, and pumps.

This book is the result of a careful selection of contributors in the field of CFD. It is divided into three sections according to the purpose and approaches used in the development of the contributions. The first section describes the "high-performance computing" (HPC) tools and their impact on CFD modeling. The second section is dedicated to "CFD models for local and large-scale industrial phenomena." Two types of approaches are basically contained here: one concerns the adaptation from global to local scale, - e.g., the applications of CFD to study the climate changes and the adaptations to local scale. The second approach, very challenging, is the multiscale analysis. The third section is devoted to "CFD in numerical modeling approach for experimental cases." Its chapters emphasize on the numerical approach of the mathematical models associated to few experimental (industrial) cases. Here, the impact and the importance of the mathematical modeling in CFD are focused on. It is expected that the collection of these chapters will enrich the state of the art in the CFD domain and its applications in a lot of fields. This collection proves that CFD is a highly interdisciplinary research area, which lies at the interface of physics, engineering, applied mathematics, and computer science.

Advances in Renewable Energies Offshore is a collection of the papers presented at the 3rd International Conference on Renewable Energies Offshore (RENEW 2018) held in Lisbon, Portugal, on 8-10 October 2018. The 104 contributions were written by a diverse international group of authors and have been reviewed by an International Scientific Committee. The book is organized in the following main subject areas: - Modelling tidal currents - Modelling waves - Tidal energy devices (design, applications and experiments) - Tidal energy arrays - Wave energy devices (point absorber, multibody, applications, control, experiments, CFD, coastal OWC, OWC and turbines) - Wave energy arrays - Wind energy devices - Wind energy arrays - Maintenance and reliability - Combined platforms - Moorings, and - Flexible materials Advances in Renewable Energies Offshore collects recent developments in these fields, and will be of interest to academics and professionals involved in the above mentioned areas.

In recent years, the sustainability and safety of perishable foods has become a major consumer concern, and refrigeration systems play an important role in the processing, distribution, and storage of such foods. To improve the efficiency of food preservation technologies, it is necessary to explore new technological and scientific advances both in materials and processes. The Handbook of Research on Advances and Applications in Refrigeration Systems and Technologies gathers state-of-the-art research related to thermal performance and energy-efficiency. Covering a diverse array of subjects—from the challenges

of surface-area frost-formation on evaporators to the carbon footprint of refrigerant chemicals—this publication provides a broad insight into the optimization of cold-supply chains and serves as an essential reference text for undergraduate students, practicing engineers, researchers, educators, and policymakers.
This book presents a collection of papers emphasizing applications of mathematical models and methods to real-world problems of relevance for industry, life science, environment, finance and so on. The biannual Conference of ECMI (the European Consortium of Mathematics in Industry) held in 2014 focused on various aspects of industrial and applied mathematics. The five main topics addressed at the conference were mathematical models in life science, material science and semiconductors, mathematical methods in the environment, design automation and industrial applications, and computational finance. Several other topics have been treated, such as, among others, optimization and inverse problems, education, numerical methods for stiff pdes, model reduction, imaging processing, multi physics simulation, mathematical models in textile industry. The conference, which brought together applied mathematicians and experts from industry, provided a unique opportunity to exchange ideas, problems and methodologies, bridging the gap between mathematics and industry and contributing to the advancement of science and technology. The conference has included a presentation of EU-Maths-In (European Network of Mathematics for Industry and Innovation), a recent joint initiative of ECMI and EMS. The proceedings from this conference represent a snapshot of the current activity in industrial mathematics in Europe, and are highly relevant to anybody interested in the latest applications of mathematics to industrial problems.
This collection explores computational fluid dynamics (CFD) modeling and simulation of engineering processes, with contributions from researchers and engineers involved in the modeling of multiscale and multiphase phenomena in material processing systems. The papers cover the following processes: Iron and Steelmaking (Tundish, Casting, Converter, Blast Furnace); Microstructure Evolution; Casting with External Field Interaction; and Smelting, Degassing, Ladle Processing, Mechanical Mixing, and Ingot Casting. 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.
Inhaltsangabe:Abstract: Wind energy is an increasingly import source of renewable, clean energy. In spite of this, only the methods and the materials of construction have improved over time, while the basic working principle of the wind turbine is still the same as it was centuries ago. In this thesis we have increased the power of a wind turbine by a factor of 4 in a fluid dynamic simulation, using a very simple external shroud system. We have also extended the theory of wind turbines (limit of Betz), to include this new kind of device and show why past attempts to augment the power of a wind turbine by means of shroud systems have failed. A detailed analysis of the device and its functioning principle is presented in this thesis - optimization studies need to be done in the future. Inhaltsverzeichnis:Table of Contents:

The present study analyzes the drift velocity of a synthetic oil in horizontal two-phase slug flow pipelines, by evaluating the effect of some physical properties, such as density and dynamic viscosity, and pipeline characteristics, such as the length of the pipe, due to its applications in various industries as in the O&G industry processes. This was achieved by using Computational Fluid Dynamics (CFD) tool approaches. The STAR-CCM+ software was utilized to simulate a half circular pipeline with a symmetry plane in a 2D mesh model, analyzing three different turbulence models. This model was fixed with a mesh independence test to obtain an accurate number of cells for the grid. The CFD results were compared with the experimental data gathered by the Tulsa University Fluid Flow Project (2018) research group. The drift velocity results achieved with a reasonable accuracy level in the pipeline, with error values under 15%. A dimensionless analysis for the experimental and CFD Reynolds numbers was done, concluding that the drift velocity within the pipe is dominated by viscous forces that overcome the inertial forces.

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.

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.

The book contains the research contributions belonging to the Special Issue "Numerical Simulation of Wind Turbines", published in 2020-2021. They consist of 15 original research papers and 1 editorial. Different topics are discussed, from innovative design solutions for large and small wind turbine to control, from advanced simulation techniques to noise prediction. The variety of methods used in the research contributions testifies the need for a holistic approach to the design and simulation of modern wind turbines and will be able to stimulate the interest of the wind energy community.

Together with turbulence, multiphase flow remains one of the most challenging areas of computational mechanics and experimental methods and numerous problems remain unsolved to date. Multiphase flows are found in all areas of technology, at all length scales and flow regimes. The fluids involved can be compressible or incompressible, linear or nonlinear. Because of the complexity of the problems, it is often essential to utilize advanced computational and experimental methods to solve the complex equations that describe them. Challenges in these simulations include modelling and tracking interfaces, dealing with multiple length scales, modelling nonlinear fluids, treating drop breakup and coalescence, characterizing phase structures, and many others. Experimental techniques, although expensive and difficult to perform, are essential to validate models. This book contains papers presented at the Fifth International Conference on Computational Methods in Multiphase Flow, which are grouped into the following topics: Multiphase Flow Simulation; Interaction of Gas, Liquids and Solids; Turbulent Flow; Environmental Multiphase Flow; Bubble and Drop Dynamics; Flow in Porous Media; Heat Transfer; Image Processing; Interfacial Behaviour.

Sediment transport is a book that covers a wide variety of subject matters. It combines the personal and professional experience of the authors on solid particles transport and related problems, whose expertise is focused in aqueous systems and in laboratory flumes. This includes a series of chapters on hydrodynamics and their relationship with sediment transport and morphological development. The different contributions deal with issues such as the sediment transport modeling; sediment dynamics in stream confluence or river diversion, in meandering channels, at interconnected tidal channels system; changes in sediment transport under fine materials, cohesive materials and ice cover; environmental remediation of contaminated fine sediments. This is an invaluable interdisciplinary textbook and an important contribution to the sediment transport field. I strongly recommend this textbook to those in charge of conducting research on engineering issues or wishing to deal with equally important scientific problems.

This paper describes the development of a computational fluid dynamics (CFD) methodology to simulate the hydrodynamics of horizontal-axis tidal current turbines. Qualitative measures of the CFD solutions were independent of the grid resolution. Conversely, quantitative comparisons of the results indicated that the use of coarse computational grids results in an under prediction of the hydrodynamic forces on the turbine blade in comparison to the forces predicted using more resolved grids. For the turbine operating conditions considered in this study, the effect of the computational timestep on the CFD solution was found to be minimal, and the results from steady and transient simulations were in good agreement. Additionally, the CFD results were compared to corresponding blade element momentum method calculations and reasonable agreement was shown. Nevertheless, we expect that for other turbine operating conditions, where the flow over the blade is separated, transient simulations will be required.

The exploitation of small horizontal axis wind turbines provides a clean, prospective and viable option for energy supply. Although great progress has been achieved in the wind energy sector, there is still potential space to reduce the cost and improve the performance of small wind turbines. An enhanced understanding of how small wind turbines interact with the wind turns out to be essential. This work investigates the aerodynamic design and analysis of small horizontal axis wind turbine blades via the blade element momentum (BEM) based approach and the computational fluid dynamics (CFD) based approach. From this research, it is possible to draw a series of detailed guidelines on small wind turbine blade design and analysis. The research also provides a platform for further comprehensive study using these two approaches. The wake induction corrections and stall corrections of the BEM method were examined through a case study of the NREL/NASA Phase VI wind turbine. A hybrid stall correction model was proposed to analyse wind turbine power performance. The proposed model shows improvement in power prediction for the validation case, compared with the existing stall correction models. The effects of the key rotor parameters of a small wind turbine as well as the blade chord and twist angle distributions on power performance were investigated through two typical wind turbines, i.e. a fixed-pitch variable-speed (FPVS) wind turbine and a fixed-pitch fixed-speed (FPFS) wind turbine. An engineering blade design and analysis code was developed in MATLAB to accommodate aerodynamic design and analysis of the blades. The linearisation for radial profiles of blade chord and twist angle for the FPFS wind turbine blade design was discussed. Results show that, the proposed linearisation approach leads to reduced manufacturing cost and higher annual energy production (AEP), with minimal effects on the low wind speed performance. Comparative studies of mesh and turbulence models in 2D and 3D CFD modelling were conducted. The CFD predicted lift and drag coefficients of the airfoil S809 were compared with wind tunnel test data and the 3D CFD modelling method of the NREL/NASA Phase VI wind turbine were validated against measurements. Airfoil aerodynamic characterisation and wind turbine power performance as well as 3D flow details were studied. The detailed flow characteristics from the CFD modelling are quantitatively comparable to the measurements, such as blade surface pressure distribution and integrated forces and moments. It is confirmed that the CFD approach is able to provide a more detailed qualitative and quantitative analysis for wind turbine airfoils and rotors. With more advanced turbulence model and more powerful computing capability, it is prospective to improve the BEM method considering 3D flow effects.

This book presents the latest research in one of the most challenging, yet most universally applicable areas of technology. Multiphase flows are found in all areas of technology, at all length scales and flow regimes, involving compressible or incompressible linear or nonlinear fluids. The range of related problems of interest is vast, including astrophysics, biology, geophysics, atmospheric process, and many areas of engineering. The solution of the equations that describe such complex problems often requires a combination of advanced computational and experimental methods. For example, any models developed must be validated through the application of expensive and difficult experimental techniques. Numerous problems in the area thus remain as yet unsolved, including modelling nonlinear fluids, modelling and tracking interfaces, dealing with multiple length scales, characterising phase structures, and treating drop break-up and coalescence. The papers contained in the book were presented at the eighth in a well established series of biennial conferences that began in 2001. They represent close interaction between numerical modellers and other researchers working to gradually resolve the many outstanding issues in understanding of multiphase flow. The papers in the book cover such topics as: Multiphase Flow Simulation; Bubble and Drop Dynamics; Interface Behaviour; Experimental Measurements; Energy Applications; Compressible Flows; Flow in Porous Media; Turbulent Flow; Image Processing; Heat Transfer; Atomization; Hydromagnetics; Plasma; Fluidised Beds; Cavitation.

This book addresses selected topics in electrical engineering, electronics and mechatronics

that have posed serious challenges for both the scientific and engineering communities in recent years. The topics covered range from mathematical models of electrical and electronic components and systems, to simulation tools implemented for their analysis and further developments; and from multidisciplinary optimization, signal processing methods and numerical results, to control and diagnostic techniques. By bridging theory and practice in the modeling, design and optimization of electrical, electromechanical and electronic systems, and by adopting a multidisciplinary perspective, the book provides researchers and practitioners with timely and extensive information on the state of the art in the field — and a source of new, exciting ideas for further developments and collaborations. The book presents selected results of the XIII Scientific Conference on Selected Issues of Electrical Engineering and Electronics (WZEE 2016), held on May 04–08, 2016, in Rzeszów, Poland. The Conference was organized by the Rzeszów Division of Polish Association of Theoretical and Applied Electrical Engineering (PTETiS) in cooperation with the Faculty of Electrical and Computer Engineering of the Rzeszów University of Technology.
Thermal processes are key manufacturing steps in producing durable and useful products, with solidification, welding, heat treating, and surface engineering being primary steps. These papers represent the latest state-of-the-art in thermal process modeling. The breadth of topics covers the depth of the industry.
Advances of Computational Fluid Dynamics in Nuclear Reactor Design and Safety Assessment presents the latest computational fluid dynamic technologies. It includes an evaluation of safety systems for reactors using CFD and their design, the modeling of Severe Accident Phenomena Using CFD, Model Development for Two-phase Flows, and Applications for Sodium and Molten Salt Reactor Designs. Editors Joshi and Nayak have an invaluable wealth of experience that enables them to comment on the development of CFD models, the technologies currently in practice, and the future of CFD in nuclear reactors. Readers will find a thematic discussion on each aspect of CFD applications for the design and safety assessment of Gen II to Gen IV reactor concepts that will help them develop cost reduction strategies for nuclear power plants. Presents a thematic and comprehensive discussion on each aspect of CFD applications for the design and safety assessment of nuclear reactors Provides an historical review of the development of CFD models, discusses state-of-the-art concepts, and takes an applied and analytic look toward the future Includes CFD tools and simulations to advise and guide the reader through enhancing cost effectiveness, safety and performance optimization
Computational fluid dynamics (CFD), which uses numerical analysis to predict and model complex flow behaviors and transport processes, has become a mainstream tool in engineering process research and development. Complex chemical processes often involve coupling between dynamics at vastly different length and time scales, as well as coupling of different physical models. The multiscale and multiphysics nature of those problems calls for delicate modeling approaches. This book showcases recent contributions in this field, from the development of modeling methodology to its application in supporting the design, development, and optimization of engineering processes.
Slurry Flow: Principles and Practice describes the basic concepts and methods for understanding and designing slurry flow systems, in-plan installations, and long-distance transportation systems. The goal of this book is to enable the design or plant engineer to derive the maximum benefit from a limited amount of test data and to generalize operating experience to new situations. Design procedures are described in detail and are accompanied by illustrative examples needed by engineers with little or no previous experience in slurry transport. The technical literature in this field is extensive: this book facilitates its use by surveying current research results and providing explanations of mechanistic flow models. This discussion of background scientific principles helps the practitioner to better interpret test data,

select pumps, specify materials of construction, and choose measuring devises for slurry transport systems. The extensive range of topics covered in Slurry Flow: Principles and practice includes slurry rheology, homogeneous and heterogeneous slurry flow principles, wear mechanisms, pumping equipment, instrumentation, and operating aspects.
Copyright: 79f45227e5673094a6dd68dc682d3007