Cfd Modelling Of A Horizontal Three Phase Separator A

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.

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 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.

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.

Computational Fluid Dynamic Analysis of Horizontal Axis Wind Turbine
Computational Fluid Dynamics Simulations
CFD Modelling of Turbulent Flow in Open-Channel Expansions

Numerical Simulation of Wind Turbines

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.

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.

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.

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.

Development and Verification of a Computational Fluid Dynamics Model of a Horizontal-Axis Tidal Current Turbine

CFD Analysis of Unsteady Hydrodynamic Loading on Horizontal Axis Tidal Turbine (HATT) Blades
Sediment Transport

An Overview of Heat Transfer Phenomena

Ventilation for Health and Sustainable Environment

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.

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.

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.

In this Special Issue, one review paper highlights the necessity of multiscale CFD, coupling micro- and macro-scales, for exchanging information at the interface of the two scales. Four research papers investigate the hydrodynamics, heat transfer, and chemical reactions of various processes using Eulerian CFD modeling. CFD models are attractive for industrial applications. However, substantial efforts in physical modeling and numerical implementation are still required before their widespread implementation.

Air Distribution in Rooms

Volume 2

Multiscale and Multiphysics Challenges

Thermal Process Modeling 2014:

Computational Fluid Dynamics (CFD) of Chemical Processes

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.

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.

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.

Horizontal Axis Tidal Turbines (HATTs) can experience amplified, time varying hydrodynamic loads during operation due to dynamic stall. Elevated hydrodynamic loads impose high structural loads on turbine blades, thus appreciably shortening machine service life. An improved characterization of the unsteady hydrodynamic loads on tidal turbine blades is therefore necessary to enable more reliable predictions of their fatigue life and to avoid premature failures. This thesis reports on a Computational Fluid Dynamics (CFD) analysis of the unsteady blade loading of a scale-model HATT taking dynamic stall into account. Numerical simulations are performed both in two-dimensional (2-D) and three-dimensional (3-D) using the commercial CFD solver ANSYS Fluent.After a brief description of the theories and methods involved, the behaviour of flow at low Reynolds number around a NACA-0012 aerofoil pitching in a sinusoidal pattern that induces dynamic stall is studied firstly to validate the numerical method and the choice of turbulence models. Then full 3-D computations of a rotating scale-model HATT rotor are presented for steady and periodic unsteady inflow situations, respectively. The reliability of the 3-D numerical method is evaluated by comparing the blade loads, especially the out-of-plane blade-root bending moment (defined as being about an axis normal to the rotor axis), with measurement data obtained from experimental tests conducted at the University of Strathclyde’s Kelvin Hydrodynamics Laboratory towing tank. Analyses in the steady velocity study are documented for a broad range of rotor speeds and flow velocities. Furthermore, investigations of 3-D flow separation and scale effects on blade loads are also performed.The periodic unsteady velocity study aims to examine the out-of-plane blade-root bending moment response to harmonic axial motion, deemed representative of the free-stream velocity perturbations induced by the unsteady flow. Parametric tests on oscillatory frequencies and amplitudes are carried out in order to analyse the HATT blade hydrodynamic behaviour under different flow patterns. Detailed flow field data is analysed to understand 3-D dynamic stall from a modelling perspective.It is concluded that the results by the present study provide significant insights into the flow physics occurring around the HATT rotor blades under various flow conditions. The CFD method can be used for designing more advanced HATT rotors, it also can be used to fine tune the computationally faster lower order Blade Element Momentum (BEM) methods for parametric design studies where experimental data is not available, particularly at the challenging rotor operating conditions involving flow separation and dynamically varying hydrodynamic behaviours.

The Study of Two-phase Flow for Crude Oil Production Using Computational Fluid Dynamics (CFD)

Computational Methods in Multiphase Flow V

Volume 2: Numerical Modelling in Mechanical and Materials Engineering, NME 2018, 28-29 August 2018, Ghent University, Belgium

Principles and Practice

Analysis and Simulation of Electrical and Computer Systems

The present study analyzes the influence of the physical properties of Newtonian and non-Newtonian fluids, such as density, effective viscosity and surface tension, as well as operational parameters of the piping, such as diameter, length and inclination angle on the drift velocity for two-phase gas-liquid flow. This study comprises experimental and Computational Fluid Dynamic (CFD) approaches. The simulation model was calibrated through a mesh independence test, which considered experimental and literature data as benchmark, and was divided in three sections based on the viscosity of the fluid and the positioning of the pipeline. The results obtained through the CFD model showed good agreement with the experimental data gathered for the present study, keeping the experimental deviations under 30% for all cases. The relationship between the Froude number (Fr) and Viscosity number (Nvis) was studied and an inverse exponential pattern was found for all the parameters and fluids tested. Additionally, the relation observed between Fr and Nvis for a horizontal setting on Newtonian fluids had good resemblance with the models proposed in literature, predicting accurately a decreasing behavior of the drift velocity for increasing viscosity. The data gathered for all fluids on the drift velocity's behavior against operational parameters showed the influence of the governing forces for each case based on a dimensionless analysis using Etvos (Eo) and Reynolds (Re) numbers. For dominant capillary or viscous forces, the drift velocity changed with the variation of these parameters. However, it was found that for dominant inertial and gravitational forces, the drift velocity maintained a constant value regardless of the operational settings. Finally, it was observed that the rheological nature of the non-Newtonian fluids has a significant influence on the drift velocity's behavior, deviating its patterns from the Newtonian fluids as the effective viscosity changes.

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.

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.

Slug flow can emerge as a factor in several industrial processes, especially in the oil and gas (O&G) industry. However, because of the complications that are inherent in multiphase flow, finding or developing a viable analysis tool has thus far proven elusive. For the past few decades, computational fluid dynamics (CFD) has served as the preferred approach in the flow analysis of single phase flow, yet it is only now beginning to be used in multiphase flow. Therefore, if CFD is to be adopted on a larger scale in the

(O&G) industry, it is imperative that we first explore the wide variety of models currently existing in the commercial realm. This thesis investigates the commercial CFD package ANSYS 16.2 analysis of (air-water slug flow) and (water-sand slurry flow) inside a horizontal pipe (2-15 m long with a 0.05-0.059 m internal diameter) and an annular pipe (2- 4.5 m long, 0.02- 0.088 m inner and 0.04-0.12 m outer diameter). A range of two-phase air/water flow simulations is carried out using the Eulerian model with the Reynolds Stress Model (RSM), and the volume of fluid (VOF) model with the Shear-Stress-Transport (SST) model option of turbulence. The aim is to simulate a range of fluid velocities between 1.66 and 7 m/s and a range of gas velocities between 0.55 and 11 m/s. Additional investigations include comparing CFD predictions along with experimental measurements in the literature and performing sensitivity studies. In the present work, the impact from fluid-structure interaction (FSI) is demonstrated by using a 2-way coupling in ANSYS, effectively joining CFD and structural analysis. At the same time, stress and pressure changes are measured, along with the deformational structural response arising from unsteady multiphase flow. It is hoped that the outcome of this study will assist engineers and researchers in making better decisions in terms of operation, design, and sizing of two-phase flow systems, as these systems have broad and promising applications in subsea (O&G) pipelines.

Sustainable Energy-Water-Environment Nexus in Deserts

CFD Simulation of Annular Flows Through Bends

Proceedings from the Fifth International Conference on Thermal Process Modeling and Computer Simulation

Proceedings of the 1st International Conference on Numerical Modelling in Engineering

Engineering Fluid Dynamics 2019-2020

This book addresses challenges and opportunities in the Energy-Water-Environment (EWE) nexus, with a particular focus on research and technology development requirements in harsh desert climates. Its chapters include selected contributions presented during the 1st international conference on sustainable Energy-Water-Environment nexus in desert climates (ICSEWEN-19) held at the Qatar Environment and Energy Research Institute (QEERI) in Doha, Qatar in December 2019. This volume is comprised of three main chapters, each describing important case studies and progress on water, energy and environmental questions. A fourth chapter on policies and community outreach on these three areas is also included. This compilation aims to bridge the gap between research and industry to address the socioeconomic impacts of the nexus imbalance as perceived by scientists, industrial partners, and policymakers. The content of this book is of particular importance to graduate students, researchers and decision makers interested in understanding water, energy and environmental challenges in arid areas. Re searchers in environmental and civil engineering, chemistry, hydrology and environmental science can also find unique in-situ observations of the current nexus imbalance in deserts climate to validate their investigations. It is also an invaluable guide for industry professionals working in water, energy, environment and food sectors to understand the rapidly evolving landscape of the EWE nexus in arid areas. The analyses, observations and lessons-learned summarized herein are applicable to other arid areas outside North Africa and the Arabian Peninsula as well, such as central Australia, the southwest of the United States and deserts in central Asia.

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.

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.

There is particular interest in the oil industry, in gas/liquid distribution in pipe line systems. The presence of bends has a significant effect on gas/liquid flows. Bends are often necessary to fit the equipment into limited spaces e.g. in plants or on oil rig platforms. As part of designing industrial systems, it is therefore important to be able to understand how liquid and gas move around bends. The aim of this research is to develop a method for predicting gas/liquid annular flows. A 3D CFD-based method is therefore developed to solve for annular flows in pipes and is applied to a range of pipe bends. In the presented study, the two-phases are gas and liquid. Multiphase fields can be handled as a continuum gas field, continuum liquid filed and as liquid droplets of varied diameters. The liquid travels along the walls as a film and in the gas core in the form of droplets. The presented approach accounts for the dynamics of the droplets flow in the gas core and their interaction between them. The liquid film is solved explicitly by means of a modified Volume of Fluid (VOF) method. The droplets are traced using a Lagrangian technique. The film to droplets (entrainment) and droplets to film (splashing, spread, bounce and stick) interactions are taken into account using sub- models to complement the VOF model. In free surface flows, a high velocity gradient at the gas/liquid interface results in high turbulence generation. In order to improve the momentum transfer between the phases at the interface, a correction to VOF is also implemented based on the work of Egorov [1]. A detailed comparison between the model and experimental data for vertical, Wolf et al. [2], and horizontal annular flows, Butterworth and Pulling [3], show reasonable agreement. The model is then applied to annular flow in bends, Maddock et al. [4], Anderson and Hills [5], Sakamoto et al. [6]. The comparison between the model and experimental data found in the literature show a good agreement. The model is also successfully applied to medium size (127mm) pipe configurations run at Nottingham University as part of a parent project. The model is finally applied to large pipe diameters encountered in industrial oil/gas applications to investigate scale issues and the model potential in industry.

Computational Methods in Multiphase Flow VII

A Comparison of Experimental Study, Empirical Correlation, CFD Simulation and Numerical Model in Prediction of Cuttings Bed Height in Horizontal Wells

CFD Modelling of Fluid Hydrodynamics in Horizontal Primary Separator

Two-Dimensional Computational Fluid Dynamics and Conduction Simulations of Heat Transfer in Horizontal Window Frames with Internal Cavities

Advances of Computational Fluid Dynamics in Nuclear Reactor Design and Safety Assessment

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

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.

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

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.

Progress in Industrial Mathematics at ECMI 2014

The Proceedings of the 5th Asia-Pacific Drying Conference

Computational Fluid Dynamics (CFD) to Simulate Slug Flow in Horizontal Pipeline and Annular Pipe

Design of solid-liquid systems

Handbook of Research on Advances and Applications in Refrigeration Systems and Technologies

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.

Proceedings of a symposium sponsored by Association for Iron and Steel Technology and the Process Technology and Modeling Committee of the Extraction and Processing Division and the Solidification Committee of the Materials Processing and Manufacturing Division of TMS (The Minerals, Metals & Materials Society) Held during the TMS 2012 Annual Meeting & Exhibition Orlando, Florida, USA, March 11-15, 2012

CFD Modelling of Fluid Hydrodynamics in Horizontal Primary SeparatorThe Study of Two-phase Flow for Crude Oil Production Using Computational Fluid Dynamics (CFD)

Experimental and CFD Modelling of the Drift Flux in Two-phase Air-(non)Newtonian Slug-flow Pattern Flow Along Horizontal and Inclined Pipelines

Proceedings of the 3rd International Conference on Renewable Energies Offshore (RENEW 2018), October 8-10, 2018, Lisbon, Portugal

Handbook of Industrial Drying

2D CFD Modelling of the Drift Flux Velocity in Two-phase Air-Newtonian Slug-flow Pattern Flow Along Horizontal Pipelines

CFD Modeling of Complex Chemical Processes