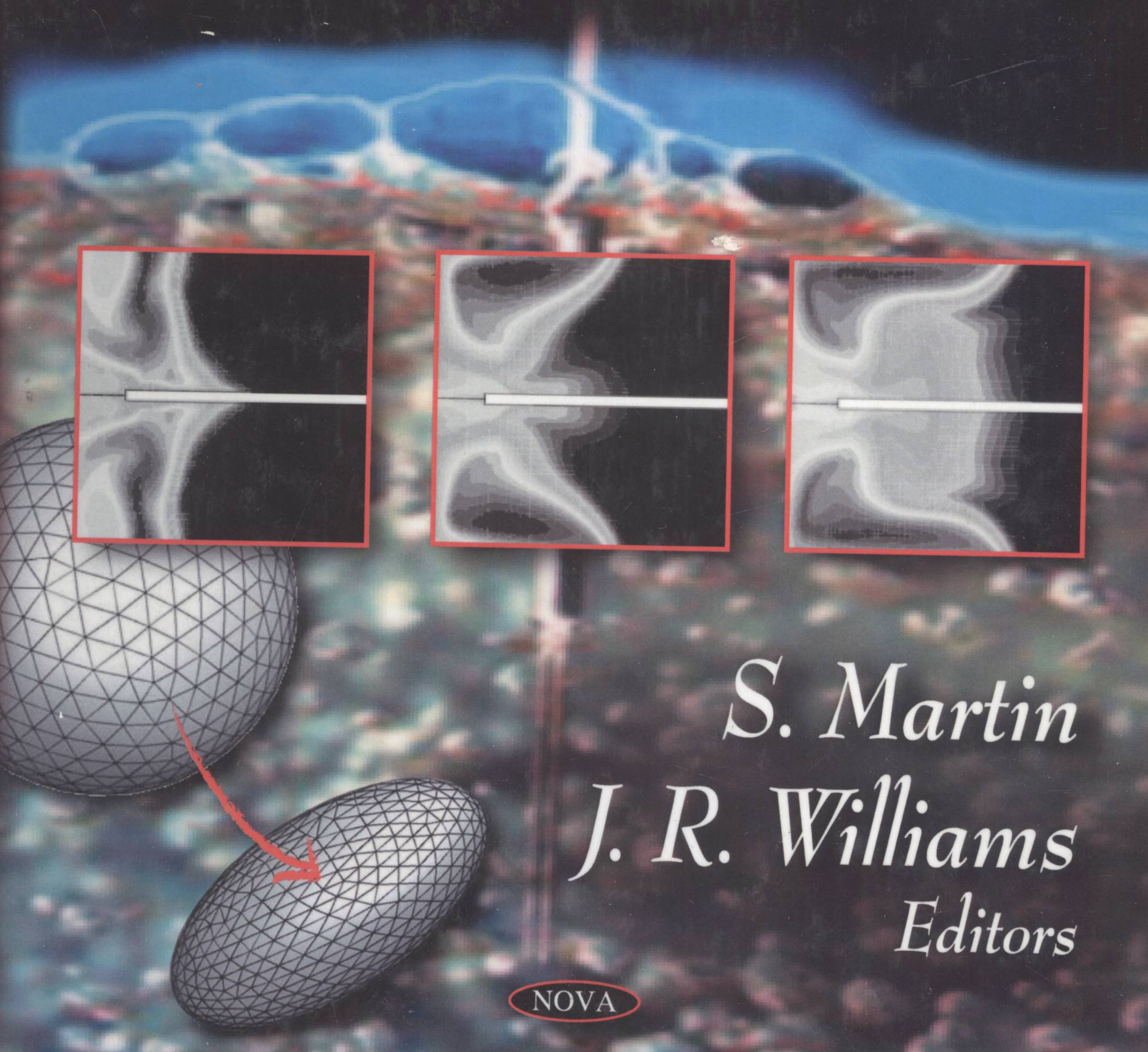
Multiphase Flow Research



S. Martin
J. R. Williams
Editors

NOVA

MULTIPHASE FLOW RESEARCH

S. MARTIN
AND
J.R. WILLIAMS
EDITORS

Nova Science Publishers, Inc.
New York

For permission to use material from this book please contact us:
Telephone 631-231-7269; Fax 631-231-8175
Web Site: http://www.novapublishers.com

LIBRARY OF CONGRESS CATALOGING-IN-PUBLICATION DATA
Available upon request.

ISBN 978-1-60692-448-8

Published by Nova Science Publishers, Inc. ✦ New York

MULTIPHASE FLOW RESEARCH

PREFACE

In fluid mechanics, multiphase flow is a generalisation of the modelling used in two-phase flow to cases where the two phases are not chemically related or where more than two phases are present. Each of the phases is considered to have a separately defined volume fraction , and velocity field. Conservation equations for the flow of each species, can then be written down straightforwardly.The momentum equation for each phase is less straightforward. It can be shown that a common pressure field can be defined, and that each phase is subject to the gradient of this field, weighted by its volume fraction. Transfer of momentum between the phases is sometimes less straightforward to determine, and in addition, a very light phase in bubble form has a virtual mass associated with its acceleration. This new book presents the latest research in the field.

Within Chapter 1 simulation models to describe multiphase flow systems with are presented and discussed with respect to their physical background, their application range and their numerical implementation. The models comprise multiphase flow systems with different phase combinations and for different phase concentrations. Depending on the models multiphase flow systems can be described in an integral character neglecting details in the transport of single particles but also in a very detailed, microscopic manner with a close look on the single particle behavior. Besides the discussion of well-established models (Euler-Lagrange and Euler-Euler models), new developments are described which allow the simulation of volumetrically resolved particles, the calculation of the dynamic of phase interfaces and the inclusion of particle-particle interaction mechanisms like agglomeration, coalescence or particle breakup.

Multiphase flow is used to refer any fluid flow consisting of more than one phase. The constituent fluids usually have different physical properties separated by an interface that evolves with the flow. Application of multiphase flows in different industries and research fields are enormous. A good understanding of underlying physics of multiphase flows is the need to model and predict the detailed behavior of those flows. Although efforts to compute the motion of multiphase flows are as old as computational fluid dynamics, the difficulty in solving the full Navier–Stokes equations in the presence of a deforming interface and recognition of the interface at every instant of time is a real challenge. The phase change heat transfer associated with multiphase flow is very common in several engineering applications and adds more complexity in the modeling of multiphase flow. In Chapter 2 an overview of multiphase flow modeling techniques for different applications are reviewed. Macroscopic, mesoscopic and microscopic modeling techniques for multiphase flow are discussed with respect to grid based and particle based methods. A special attention has been given to the grid based interface capturing methods which are best suited for modeling multiphase flow with phase change heat transfer.

The computational modeling efforts so far in phase change heat transfer applications are reviewed in this article in great details. Some of them include bubble dynamics in nucleate boiling, flow boiling and film boiling; droplet and surface interaction and droplet and bubble dynamics in thin liquid film model for spray cooling applications. The numerical and computational issues and challenges involved in modeling these phenomena are identified. Implementation of multiphase flow modeling in parallel computing environment is also discussed. The current understanding and lack of understanding of underlying physics of bubble nucleation and phase change heat transfer from multiphase flow modeling are summarized and future directions of research areas are identified.

The need for high-performance thermal protection and fluid management techniques for systems ranging from cryogenic reactant storage devices to primary structures and propulsion systems exposed to extreme high temperatures, and other space systems such as cooling or environmental control for advanced space suits and integrated electronic circuits, requires an effective cooling system to accommodate the compact nature and high heat fluxes associated with these applications.

A two-phase forced-convection, phase-transition system can accommodate such requirements through the use of the concept of Advanced Micro Cooling Modules (AMCMs), which are essentially compact two-phase heat exchangers constructed of microchannels and designed to remove large amounts of heat rapidly from critical systems by incorporating phase transition.

It is still recognized today that two-phase flow is scientifically one of most challenging fluid dynamic problems to be explored since the 1940s, and this is further complicated when the fluid flows through channels of sub millimeter dimension. Consequently, to successfully design, analyze, and control such systems, it is necessary to first obtain a fundamental understanding of the two-phase flow and heat transfer in the microchannels. This is accomplished through the development of a quantitative model incorporating in situ concentration/void fraction, velocities, and flow patterns.

Realizing the significance of research in Chapter 3, this paper presents the results of experimental research on two-phase flow in microchannels with verification and identification of data using concomitant measurement systems. Based on the results of the experimental research conducted on air-water mixture flows in the entire range of concentration and flow patterns in a horizontal square microchannel, a mathematical model based on in situ parameters is developed and presented, which describes pressure losses in two-phase flow incorporating flow pattern phenomena. Validation of the model is accomplished using five concomitant measurement methods (capacitive and conductive concentration measurements, differential and static pressure, and optical signals). A hypothetical model for the two-phase heat transfer coefficient is also presented, which incorporates the flow patterns through the use of a flow pattern coefficient.

Chapter 4 investigates the influence of the interfacial drag on the pressure loss of combined liquid/vapour flow through porous media. This is motivated by the coolability of fragmented corium with internal heat sources, which are expected during a severe accident in a nuclear power plant. Due to the decay heat in the particles cooling water is evaporated. To reach steady states the out-flowing steam must be replaced by in-flowing water. The pressure field inside the porous structure determines the water ingression, and in effect the overall coolability.

Typically, correlations for the dryout heat flux of porous media are adjusted to measurements where water ingression is from a pool positioned above. In these models the nature of the two-phase flow is included in corrections of the permeability and passability, achieved by simple functions of the void fraction. However, already configurations with possible water ingression from below demonstrate that this treatment is insufficient. The drag between the liquid and the vapour phase supports the co-current water inflow from below, and hinders the counter-current flow from above. Thus, the interfacial drag must be explicitly included in the modeling.

This necessity is already seen in the measured pressure loss of simple isothermal air/water flow in porous media as well as for boiling particle beds with water ingression from below. Based on such experiments, two models with explicit consideration of the interfacial drag from the literature are discussed. Different flow patterns for the drag coefficients are included with the most advanced of these models. This article proposes some modifications on this model with respect to the small particle sizes that are expected in the reactor application. Furthermore, modifications to the formulation in the annular flow regime are necessary, as here the original model yields unreliable results.

Based on the friction laws referred, the models are applied to typical reactor invessel and ex-vessel configurations in two-dimensional geometry. An enhanced overall coolability of the particle bed is already reached by the 2-D nature of the arrangement. Furthermore, the supporting influence of the realistic modeling by explicit consideration of the interfacial drag in the enhanced models is shown.

Fine dispersion of gas into liquid is one of the most important criteria for momentum, mass and energy transfer between the phases. It not only provides an intense mixing but also creates increased interfacial area and high mass transfer coefficient. Various types of contacting devices have been developed for achieving effective gas-liquid mixing. These may be broadly classified as co-current, counter-current and cross current systems. Among these, increased interests on co-current contacting systems have been shown because of their ability to handle high fluid flowrate without flooding, low pressure drop, higher interfacial area and mass transfer coefficients.

A review of literature on design and development of fluid-fluid cocurrent contacting equipment shows that use of liquid jet ejectors as a gas-liquid distributor in downflow bubble column is gaining in importance as it functions both as a sparger and gas entrainment device. Ejectors are devices that utilize the kinetic energy of a high velocity liquid jet in order to entrain and disperse the gas phase. The bubble column with ejector system is very simple in design and no extra energy is required for gas dispersion as the gas phase is sucked and dispersed by the high velocity liquid jet. Thus, cocurrent down flow bubble column with ejector type gas distributor possesses the following distinct advantages over more conventional devices, such as (i) lower power consumption (ii) almost complete gas utilization (iii) higher overall mass transfer coefficient and (iv) tolerance to particulates and therefore useful for chemical reaction with slurries.

In Chapter 5 some important hydrodynamic characteristics on gas entrainment by a high velocity liquid jet; gas-holdup and bubble size distribution; two-phase frictional pressure drop; energy dissipation in ejector and contactor; interfacial area and mass transfer coefficient in an ejector induced downflow bubble column have been discussed. Further since the processing media for many processes such as sewage sludge, microbiological culture, polymer solutions etc. are non-Newtonian in nature; an attention has been paid to study with non-Newtonian fluid using Carboxy Methyl Cellulose (CMC) solution at different concentrations.

The experimental set-up consists of a column, an ejector at the top of the column and a gas-liquid separator at the bottom along with other accessories like pumps, flow meters, valves and a number of manometers connected at different heights of the column. The column and ejector assembly were fitted with perfect alignment to obtain an axially symmetric jet. Experimentally it has been found that the rate of gas entrainment is strongly dependent on the motive fluid flowrate and the pressure in the separator. A precise idea of gas holdup and bubble size distribution was obtained by measuring the column pressure readings at different points of the column. The interfacial mass transfer area and liquid side volumetric mass transfer co-efficient were measured by both physical and chemical methods. Significantly higher values of interfacial area and volumetric mass transfer coefficient are obtained at low gas flowrate in the present system and the results are compared with the other reported works.

Chapter 6 considers what happens when a multi-component system is subjected to a flow. More precisely, how to model it to better understand it. Two state of the art techniques for modeling multi-component flow are presented, based on the type of multi-component is considered. The first technique is to model a large amount of non-coalescing fluid droplets in flow, all of which have possibly different parameters (surface tensions, viscosities, radius, etc.). The second technique is to model a large amount of deformable cells or vesicles in flow, all of which have individual sets of parameters such as shape, rest curvature, elasticity, elasticity response, volume,etc. The underlying fluid dynamics of these two techniques is based on the lattice Boltzmann method, which was chosen for its efficiency and great versatility compared to other techniques. The chapter begins by deriving it, showing how it relates to more commonly known methods. The bulk of the chapter consists of proof of concepts and engineering applications for each technique, in order to demonstrate their efficiency, accuracy, versatility and potential.

Two main approaches exist for numerical computation of multiphase flow models. The implicit methods are efficient, yet inaccurate. Better accuracy is achieved by the explicit methods, which on the other hand are time-consuming.

In Chapter 7 the authors investigate generalizations of a class of *hybrid explicit-implicit* numerical schemes [SIAM J. Sci. Comput., 26 (2005), pp. 1449–1484], originally proposed for a two-fluid two-phase flow model. The authors here outline a framework for extending this class of schemes, denoted as WIMF (weakly implicit mixture flux), to other systems of conservation laws. They apply the strategy to a different two-phase flow model, the *drift-flux* model suitable for describing bubbly two-phase mixtures. Our analysis is based on a simplified formulation of the model, structurally similar to the Euler equations. The main underlying building block is a pressure-based implicit central scheme. Explicit upwind fluxes are incorporated, in a manner ensuring that upwind-type resolution is recovered for a simple contact discontinuity.

The derived scheme is then applied to the general drift-flux model. Numerical simulations demonstrate accuracy, efficiency and a satisfactory level of robustness. Particularly, it is demonstrated that the scheme outperforms an explicit Roe scheme in terms of efficiency and accuracy on slow mass-transport dynamics.

Liquid-liquid extraction columns are a quite good illustration of a two-phase flow system. They are used for contacting two immiscible liquid phases, flowing co or counter currently, and are simply vertical tubes having often, at their base, a distributor which can be a perforated plate or a series of nozzles, enabling the dispersion of one phase (the dispersed) into the other one (the continuous).

The key parameter for this type of systems is the drop sizes distribution. This latter affects directly both the hydrodynamic and the mass transfer taking place, and depends greatly on various factors particularly the influence of the flow conditions of the continuous phase on the behaviour of the dispersed phase where the dynamic of the drops is often accompanied by the breakage and/or coalescence phenomena.

In fact the study of the behaviour of the dispersed phase in a continuous flow field, generally turbulent is not easy. This is due to the nature of the macroscopic interactions generated by the breakage and coalescence phenomena resulting in a randomly distributed population of drops with respect to their sizes, concentrations and ages.

Consequently, it is necessary to consider detailed mathematical models capable of describing the events generated by the interaction between the turbulent continuous phase and the dispersed one (drops), including both phenomena *i.e.* breakage and coalescence.

Generally, the drop breakage term considers the interaction of a simple drop with the turbulent continuous phase continue turbulent, where it effectively undergoes a breakage if the transmitted turbulent kinetic energy exceeds its surface energy.

Similarly, drop coalescence can take place due to the interaction between two colliding drops in the turbulent continuous phase, and it is effective if the interstitial liquid film disposes of sufficient time to drain out .

As a first part, Chapter 8 considers the modelling of the continuous phase flow, under turbulent conditions, using the two equations model (k-ε) which has proven its ability for this type of problems. The influence of the continuous phase hydrodynamic on drop breakage and coalescence is examined, relying on the link between these phenomena and the dissipation rate of turbulent energy which can be determined from the model resolution and used for the calculation of important parameters like the maximal stable diameter, the breakage and coalescence efficiencies, etc. via computer experiments.

In the second part, this two-phase flow system is solved by the drop population balance approach where the development of many experimental research programs has greatly contributed to the elaboration of realistic models which, globally, include the drop breakage and coalescence phenomena as well as their transport. However at this level, mathematical complexities are induced and analytical solutions are not easily obtained for the drop population balance equation.

From this study one can see how complex is the modelling of the behaviour of a dispersed phase in a continuous one. The main difficulty is due to the interaction of the involved various factors as well as the nature of the problem which is rather stochastic.

Research on multiphase flows has been strongly improved over the last decades. Because of their large fields of interests and applications for chemical, hydraulic, coastal and environmental engineers and researchers, these flows have been strongly investigated. Although they are some promising and powerful numerical models and new computing tools, computations can not always solve all actual practical problems (weather forecast, wave breaking on sandy beach...). The recent and significant developments of experimental techniques such as Particle Imagery Velocimetry (PIV) and conductivity or optical probes have particularly led scientists to physical modeling that provide series of data used to calibrate numerical models. Flows with time and length scales that were not achievable in the past are now studied leading to a better description of physical mechanisms involved in mixing, diffusion and turbulence. Nevertheless, turbulence is still not well understood, particularly in two-phase flows.

In Chapter 9, the authors focus on a classical multiphase flow, the hydraulic jump. It occurs in bedrock rivers, downstream of spillways, weirs and dams, and in industrial plants. It characterizes the transition from a supercritical open-channel flow (low-depth and high velocity) to a subcritical motion (deep flow and low velocities). Experimentally, this two-phase flow can be easily studied. Furthermore, it involves fundamental physical processes such as air/water mixing and the interaction between turbulence and free surface. This flow contributes to some dissipation of the flow kinetic energy downstream of the impingement point, in a relatively short distance making it useful to minimize flood damages. It is also associated with an increase of turbulence levels and the development of large eddies with implications in terms of scour, erosion and sediment transport. These are some of the reasons that make studies on this flow particularly relevant. Although numerical and analytical studies exist, experimental investigations are still considered as the best way to improve our knowledge.

After a brief description of the hydraulic jumps, the first part of this chapter aims to review some historical developments with special regards to the experimental techniques and physical modeling (similitude). In the second part, the authors describe and discuss the basic properties of the two-phase flow including void fraction, bubble frequency, bubble velocity and bubble size. The free surface and turbulence properties are presented as well. In the last part, the authors develop some conclusions, perspectives and further measurements that should be undertaken in the future.

Chapter 10 addresses various issues related to sub- and super-critical CO_2 two-phase flow and heat transfer in macro- and micro-channels such as flow boiling, two-phase flow patterns and pressure drops without and with lubricating oil effect, supercritical gas-lubricating oil two-phase flow, two-phase flow and supercritical flow distribution in heat exchanger headers. Emphasis is given to our newly developed two-phase flow pattern map for CO_2 evaporation, flow pattern based flow boiling heat transfer and pressure drop models. Furthermore, some simulation results for electronic chips cooling using CO_2 evaporation are presented and discussed. So far, little information on CO_2 condensation is available in the literature. Therefore, this chapter does not include CO_2 condensation but it is recommended that future research be conducted in this aspect. The future research needs in CO_2 two-phase flow and heat transfer have been identified.

The proton exchange membrane (PEM) fuel cell is considered to be one of the most promising alternative clean power generators for portable, mobile and stationary applications because of its low to zero emissions, low-temperature operation, high power density and fast start-up.

Fuel (humidified hydrogen) and oxidant (humidified air) are fed into the cathode and anode, respectively. The water vapor in the cell introduced by humidification and produced by the cathode electrochemical reaction will condense to liquid water when the partial pressure of the vapor exceeds its saturation pressure. The resulting two-phase transport of water significantly affects the transport of reactants to the porous electrode and the membrane ohmic resistance, and then the cell performance. For example, liquid water accumulating in the channels and in the pores of the gas diffusion layer (GDL) increases the transport resistance of oxygen to the catalyst layer (CL), even resulting in cathode flooding at high current densities. In addition, the membrane must remain well-hydrated because the proton transport capability in the membrane is proportional to the membrane water content. Therefore, understanding and control of the water transport in the cell are important for improving cell performance. At present, the local distributions and transport of reactants in the cell are difficult to measure due to the small cell sizes; thus, numerical models (including the two-phase transport) have become an important means for understanding the reactant transport, electrochemical reactions, current density distributions, and cell performance.

Chapter 11 presents a complete three-dimensional, two-phase transport model for a PEM fuel cell based on the two-fluid method, in which the two-phase flow of the multi-component reactants and liquid water are coupled with the species transport, electrochemical reactions, proton and electron transport, electro-osmosis and back diffusion of water. Different two liquid water transport equations were developed for the various cell units based on the different liquid water transport mechanisms in the fuel cell. The model is used to investigate the effects of the various flow field designs in the bipolar plates on the cell performance. The improved understanding of the two-phase transport is then used to develop a series of new flow field designs, including the tapered flow field, contracted flow field, and partially blocked or blocked flow fields. The predictions show that the improved flow field designs significantly increase the liquid water removal from the GDL, which enhances the reactant transport rates and improves cell performance.

It is evident that in multiphase porous media, exchange of momentum, mass, and energy among the phases occurs through interfaces separating the phases. This important feature, however, is not accounted for in standard models. In fact, interfacial area does not even appear as a parameter in classical models for flow and transport in porous media. In these standard models, interphase mass transfer is most often estimated using empirical relationships.

Recently, however, new thermodynamically motivated model concepts for flow and transport in porous media have been developed which overcome this shortcoming and include interfacial areas. Preliminary modeling studies have been performed based on this new theory. They have been shown to overcome also other shortcomings of classical models like describing capillary hysteresis by employing a single function relating interfacial area, capillary pressure and saturation instead of using myriads of capillary pressure - saturation scanning curves as is the strategy of standard models.

In Chapter 12, the authors review recent experimental and numerical studies on two-phase flow in porous media that stress the need for determining interfacial areas. Also, the authors derive and propose a full set of macroscopic equations that describes two-phase flow without and with interphase mass transfer including interfacial area as a parameter. They show results of a numerical study of mass transfer between water and gas phase comparing the model including interfacial area to a standard model. It turns out that only the model with interfacial area is able to account for interphase mass transfer in a physically-based way.

In Chapter 13, an overview on experimental techniques and models applied to describe fluid dynamics in vertical conveying of coarse particles is presented, focusing on the particular characteristics obtained with the use of a spouted bed feeder as a solids feeding device. Influential factors and procedures for the prediction of regime transitions, pressure gradients and solids concentration for mechanical and non-mechanical devices are evaluated through the analysis of experimental data obtained for a wide range of experimental conditions. Finally, the experimental techniques for measuring the variables of interest and the fluid dynamic models applied to describe the flow are evaluated with particular emphasis on discussing the drawbacks and advantages of using the two-phase flow model, in analogy to single phase flow, to describe pneumatic conveying. The results were contextualized within this field in the literature, and an analysis of trends and requirements for future research is presented.

Fuel cell system is an advanced power system for the future that is sustainable, clean and environmental friendly. Small fuel cells have provided significant advantages in portable electronic applications over conventional battery systems. Competitive costs, instant recharge, and high energy density make fuel cells ideal for supplanting batteries in portable electronic devices. However, the typical proton exchange membrane (PEM) fuel cell system with its heavy reliance on subsystems for cooling, humidification and air supply would not be practical in small applications. The air-breathing PEM fuel cells without moving parts (external humidification instrument, fans or pumps) are one of the most competitive candidates for future portable-power applications.

As explained in Chapter 14, multiphase flow is a central issue in air-breathing PEM fuel cell technology because while water is essential for membrane ionic conductivity, excess liquid water leads to flooding of catalyst layers and gas diffusion layers. Understanding the flow of gas/liquid flows is therefore of major technological as well as scientific interest.

Computational fuel cell engineering tools requires the robust integration of models representing a variety of complex multi-physics transport processes characterized by a broad spectrum of length and time scales. These processes include a fascinating, but not always well understood, array of phenomena involving multiphase flow, ionic, electronic and thermal transport in concert with electrochemical reactions. The development of physically representative models that allow reliable simulation of the processes under realistic conditions is essential to the development and optimization of fuel cells, improve long-term performance and lifetime, the introduction of cheaper materials and fabrication techniques, and the design and development of novel architectures. The difficult experimental environment of fuel cell systems has stimulated efforts to develop models that could simulate and predict multi-dimensional coupled transport of reactants, multiphase flow, heat and charged species using computational fluid dynamic (CFD) methods. The strength of the CFD numerical approach is in providing detailed insight into the various flow and transport mechanisms and their interaction, and in the possibility of performing parameters sensitivity analyses.

Three-dimensional, multiphase, multi-component flow, non-isothermal CFD model of an ambient air-breathing PEM fuel cell which work in still or slowly moving air has been developed in this Chapter. The model was developed to improve fundamental understanding of the multiphase flow in a porous cathode and anode diffusion layers as well as the transport phenomena in ambient air-breathing PEM fuel cells and to investigate the impact of various operation parameters on performance. In addition to the multiphase flow and phase change in the air-breathing PEM fuel cell, the new feature of the present model is to incorporate the effect of hygro and thermal stresses into actual three-dimensional air-breathing PEM fuel cell model. Fully three-dimensional results of the multiphase velocity flow field, species profiles, liquid water saturation, temperature distribution, potential distribution, water content in the membrane, stresses distribution in the membrane and gas diffusion layers, and local current density distribution are presented and analyzed with a focus on the physical insight and fundamental understanding. They can provide a solid basis for optimizing the geometry of the PEM fuel cell stack running with a passive mode.

An overview of experimental observations of turbulence modulation in dilute multiphase flows is presented in the first Short Communication. The presence of the dispersed phase can augment or attenuate the turbulence of the continuous phase. The ratio of the dispersed phase size to the continuous phase turbulence length scale, the relative velocity of the dispersed phase with respect to the continuous phase and the Kolmogorov Stokes number affect the addition/dissipation of the continuous phase turbulence. The Kolmogorov Stokes number plays a significant role in the attenuation of turbulence. More controlled experiments are needed to isolate the effects of the dispersed phase on the turbulence energy spectrum of the continuous phase.

The main purpose of the second Short Communication is to show that the simple correlation of the turbulent viscosity as a function of column diameter only is to be understood as a correlation of eddy diffusivity in free turbulence as a function of scale only. The time-averaging procedure of Navier-Stoks equations for multiphase flow, and derivation of the theoretical distribution of time-averaged velocity, will also be briefly explained.

CONTENTS

RESEARCH AND REVIEW STUDIES