Numerical Simulation Of Two Phase Flow In An Effervescent Atomizer Numerical Simulation Of Two Phase Flow In An Effervescent Atomizer For Nano Suspension Spray

In this dissertation, we present numerical schemes in simulating immiscible two-phase flow problems. The goal of this work is to find unified solutions for numerical modeling of any kind of immiscible two-phase flow with moving interface. Fluid flows are modeled using Navier Stokes equations with discontinuous coefficients. We introduce the Volume Fraction method to evaluate discontinuous integrals arise from the variational formulation in the Finite Element method. The use of Volume Fraction method avoids the approximation of the Dirac Delta function, and therefore no regularization procedures are needed. Several operator splitting variants are studied in detail in linearized and non-linearized fashion, when we want to evaluate the discontinuous coefficients. Interface is captured using the Level Set approach, where a transport equation is solved numerically with fourth order scheme without any stabilization terms. The surface tension effect is implemented in a semi-implicit way, thus larger time steps can be used compared with the explicit method. A recent, well-developed re-initialization technique is included as a way to preserve the signed distance property of the Level Set function. All mentioned numerical methods are used to build two-dimensional solvers. Solvers have been tested both with single-phase flow and two-phase flow benchmark problems. In particular, the bubble dynamics are presented to validate stated numerical schemes.

The present thesis describes the advances made in modelling two-phase flows in inclined pipes using a transient one-dimensional approach. The research is a developement of an existing numerical methodology, capable of simulating stratified and slugging two-phase flows in horizontal or inclined single pipes. The aim of the present work is to extend the capabilities of the approach in order (i) to account for the effect of the pipe topography in the numerical solution of the two-fluid model, and (ii) to simulate vertical bubbly twophase flows at various pressures in large diameter pipes, and (iii) to model stratified and terrain-induced slugging in two-phase flow pipelines made of several uphill, downhill and level sections. A transient compressible two-fluid model based on the one-dimensional form of the mass and momentum conservation equations for the gas and liquid phases, is developed to predict those flow configurations. The wall to fluid and the interphase interactions are accounted for by constitutive relations which are flow regime dependent. The conservation equations are discretized using a finite volume method. An algorithm is created to enable simulations on pipelines made of several sections, and account for the effect of the topography in the simulations. The methodology is applied to the compressible model in order to evaluate the robustness and accuracy of the numerical schemes, especially for the high-resolution Advection Upwinding Splitting Method (AUSM) associated to the compressible model. It also assesses the ability of the method to predict three physical flow regimes, namely stratified, bubbly and terrain-induced slug flows. The terrain-induced slugging study is performed on a slightly inclined ("1.5°) V-section system. The use of hydrodynamic slug correlations for hilly-terrain slugging is discussed. It shows to be conclusive with a good agreement with experimental measurements obtained for slug frequency and slug length predictions. Mechanisms.

Spontaneous imbibition plays a very important role in the displacement mechanism of non-wetting fluid in naturally fractured reservoirs. We developed a new 2D two-phase finite element numerical model, as available commercial simulators cannot be used to model small-scale experiments with different boundary conditions as well as complex boundary conditions such as fractures and vugs. Starting with the basic equation of fluid flow, we derived the non-linear diffusion saturation equation. This equation cannot be put in weighted-integral weak variational form and hence Rayleigh-Ritz finite element method (FEM) cannot be applied. Traditionally, the way around it is to use higher order interpolation functions and use Galerkin FEM or reduce the differentiability requirement and use Mixed FEM formulation. Other FEM methods can also be used, but iterative nature of those methods makes them unsuitable for solving large-scale field problems. But if we truncate the non-linear terms and decouple the dependent variables, from the spatial as well as the temporal domains of the primary variable to solve them analytically, the non-linear FEM problem reduces to a simple weighted integral form, which can be put into its corresponding weak form. The advantage of using Rayleigh-Ritz method is that it has immediate effect on the computation time required to solve a particular problem apart from incorporating complex boundary conditions. We compared our numerical models with the analytical solution of this diffusion equation. We validated the FDM numerical model using X-Ray Tomography (CT) experimental data from the single-phase spontaneous imbibition experiment, where two simultaneously varying parameters of weight gain and CT water saturation were used and then went ahead and compared the results of FEM model to that of FDM model. A two-phase field size example was taken and results from a commercial simulator were compared to the FEM model to bring out the limitations of this approach.

Applying 3-D computer codes to isothermal and combusting two-phase flows is a multi-faceted processes. In the combustion case, this can involve the refinement of models for fuel droplet trajectories, droplet evaporation and radiation heat transfer. Turbulence modelling is also of basic importance. The realisation that many combustion flows do not satisfy the conditions of isotropy demanded by conventional models has stimulated more fundamental approaches to the Reynolds stress description. Since the common objective of such applications is to provide a basis for design, there is need for codes which are versatile, easy to operate, and interactive. This study reports some current progress in these areas. Aspects of the present code are presented including the coupling of flow and droplet motions and the interactive setting up procedure to define an arbitrary geometry. Calculated and measured velocity fields are shown for the isothermal flow in a coaxial dump combustor and results are presented for the combustion case using a baffle stabiliser.

A computer program TWOP was developed for obtaining the numerical solutions of three-dimensional, transient, two-phase flow system with nonequilibrium and nonhomogeneous conditions. TWOP employs two-fluid model and a set of the conservation equations formulated by Harlow and Amsden along with their Implicit Multi-Field (IMF) numerical technique that allows all degrees of couplings between the two fields. We have further extended the procedure of Harlow and Amsden by incorporating the implicit couplings of phase transition and interfacial heat transfer terms in the energy equations. Numerical results of two tested problems are presented to demonstrate the capabilities of the TWOP code. The first problem is the separation of vapor and liquid, showing that the code can handle the computational difficulties such as liquid packing and sharp interface phenomena. The second problem is the high pressure two-phase jet impinged on vertical plate, demonstrating the important role of the interfacial mass and momentum exchange.

This book uniquely presents an overview of methods for the numerical simulation of a wide range of two-phase flows, aimed at a broad readership of engineers and scientists at graduate level. Given that numerous methods have been proposed recently in this field, the new book series could not have been more timely and much needed for an up-to-date overview of the advances, whilst not restricting the focus on two-phase flows or any particular method. The book is written by a team of leading experts who have contributed substantially to the development of the methods and who have also applied the concepts and theories to a diverse range of applications. Computational Methods for Two-Phase Flows is self-contained, and aims to elucidate and analyze the strong relations between the various numerical methods, beyond merely giving detailed summary of the various methods. A unique enhanced feature of the book is that the sample codes that come with the book provide further benefits and ease of use to readers to implement the various numerical methods. The book is the first volume of a new book series entitled Advances in Computational Fluid Dynamics, published by World Scientific.

This book focuses on environmental sustainability by employing elements of engineering and green computing through modern educational concepts and solutions. It visualizes the potential of artificial intelligence, enhanced by business activities and strategies for rapid implementation, in manufacturing and green technology. This book covers utilization of renewable resources and implementation of the latest energy-generation technologies. It discusses how to save natural resources from depletion and illustrates facilitation of green technology in industry through usage of advanced materials. The book also covers environmental sustainability and current trends in manufacturing. The book provides the basic concepts of green technology, along with the technology aspects, for researchers, faculty, and students.

Numerical simulation of multiphase reactors with continuous liquid phase provides current research and findings in multiphase problems, which will assist researchers and engineers to advance this field. This is an ideal reference book for readers who are interested in design and scale-up of multiphase reactors and crystallizers, and using mathematical model and numerical simulation as tools. Yang and Mao’s book focuses on modeling and numerical applications directly in the chemical, petrochemical, and hydrometallurgical industries, rather than theories of multiphase flow. The content will help you to solve reacting flow problems and/or system design/optimization problems. The fundamentals and principles of flow and mass transfer in multiphase reactors with continuous liquid phase are covered, which will aid the reader’s understanding of multiphase reaction engineering. Provides practical applications for using multiphase stirred tanks, reactors, and microreactors, with detailed explanation of investigation methods. Presents the most recent research efforts in this highly active field on multiphase reactors and crystallizers. Covers mathematical models, numerical methods and experimental techniques for multiphase flow and mass transfer in reactors and crystallizers.

Keywords: two-phase flow, aerated-liquid injector, CFD.

An accurate and robust numerical method has been developed to simulate turbulent two-phase flows. The phase interface is tracked by the level-set method to capture frequent topological changes due to breaking or merging. Because of the broad-band characteristics of length scales in two-phase flow, a Lagrangian drop breakup model has been developed, which is coupled to the level-set method. In this approach, small subgrid droplets produced from resolved ligaments are then transferred from the level-set representation to the Lagrangian particles. The further secondary atomization is handled by a stochastic breakup model. When pinching-off of ligaments is not resolved on the level-set grid, a capillary breakup model is used to predict the drop size distribution from the pinching off and inserted as Lagrangian drops. This method improves the mass conservation as well as reducing the computational cost. For a high-fidelity simulation of two-phase flow, a new numerical algorithm has been developed to improve the robustness of the numerical method. The conservative formulation of Navier-Stokes equations is solved with a density correction term in the present method. The density flux terms are calculated from the level-set field for accuracy. In addition, a constant coefficient Poisson system is solved for pressure to satisfy the continuity equation in the fractional-step method. In order to show the capability of the method as an efficient tool in the breakup process, the atomization of a round liquid jet surrounded by a coaxial gas is considered. The numerical results are consistent with the observed breakup mechanisms in the experiment and the stability analysis. The drop size distribution of the resulting spray after breakup is also compared with the experimental data. The subgrid drops are also predicted by the Lagrangian drop breakup model, which shows the applicability of our method for numerical simulation of the atomization process. Both theoretical and numerical approaches are employed to investigate the stability mechanisms of the air layer drag reduction (ALDR) phenomenon. A linear viscous stability analysis is performed by solving the Orr-Sommerfeld equations in a two-dimensional two-phase Couette-Poiseuille flow configuration that mimics the far-downstream region from an air injector. Air-layer stability is reduced as the free-stream velocity, Froude number, and velocity gradients at the air-liquid interface are increased, whereas the air-layer stability is enhanced as the gas flow rate and surface tension force are increased. Nonlinear stability characteristics are also studied using numerical simulations with the same Couette flow configuration as indicated in the linear stability analysis. The study shows that the Weber number has a significant effect on the breakup of the phase interface. As the Weber number increases, the liquid ligaments become thinner, requiring higher grid resolution. Therefore, for simulations of high Weber number flows, the use of a Lagrangian spray breakup model is essential to predict the dynamics of subgrid-scale liquid structures. Direct Numerical Simulation (DNS) of two-phase flow is also performed to investigate the air layer drag reduction (ALDR) phenomenon in turbulent water flow over a backward-facing step. The Reynolds and Weber numbers based on the water properties and step height are 22,800 and 560, respectively. The total number of grid points is about 271 million for DNS. Two different air-flow injection rates are examined to investigate the mechanism and stability of the air layer. For high air-flow rate, the stable air layer is formed on the plate and more than 90% drag reduction is obtained, whereas, in the case of low air-flow rate, the air layer breaks up and ALDR is not achieved. The initial Kelvin-Helmholtz instability causes the streamwise wave structure, while turbulence interaction forms the spawise waves and causes ligament breakups. However, overall rupture of the air layer is mainly determined by the stability of the streamwise wave. The stability of the streamwise wave can be predicted from the stability analysis in the far-downstream region.

This book is the first monograph providing an introduction to and an overview of numerical methods for the simulation of two-phase incompressible flows. The Navier-Stokes equations describing the fluid dynamics are examined in combination with models for mass and surfactant transport. The book pursues a comprehensive approach: important modeling issues are treated, appropriate weak formulations are derived, level set and finite element discretization techniques are analyzed, efficient iterative solvers are investigated, implementational aspects are considered and the results of numerical experiments are presented. The book is aimed at M Sc and PhD students and other researchers in the fields of Numerical Analysis and Computational Engineering Science interested in the numerical treatment of two-phase incompressible flows.

This book is devoted to a fundamental understanding of the fluid dynamic nature of a bubble wake, more specifically the primary wake, in liquids and liquid-solid suspensions, an dto the role it plays in various important flow phenomena of multiphase systems. Examples of these phenomena are liquid/solids mixing, bubble coalescence and disintergration, particle entrainment to the freeboard, and bed contraction.

Numerical Simulation of Two-phase Flow with Front-capturing

Stochastic Tools in Turbulence discusses the available mathematical tools to describe stochastic vector fields to solve problems related to these fields. The book deals with the needs of turbulence in relation to stochastic vector fields, particularly, on three-dimensional aspects, linear problems, and stochastic model building. The text describes probability distributions and densities, including Lebesgue integration, conditional probabilities, conditional expectations, statistical independence, lack of correlation. The book also explains the significance of the moments, the properties of the characteristic function, and the Gaussian distribution from a more physical point of view. In considering fields, one must account for single-valued functions of one or more parameters, or collections of single-valued functions of one or more parameters such as time or space coordinates. The text also discusses multidimensional vector fields of finite energy, the characteristic eddies for a homogenous vector field, as well as, the distribution of solutions of an algebraic equation. Engineers, algebra students, and professors of statistics and advanced mathematics will find the book highly useful.

Escoamentos bifásicos no regime de golfadas são caracterizados pelaalternância de pacotes de líquido e grandes bolhas de gás na tubulação, sendoassociados a altas perdas de carga, além de trazer uma indesejada intermitênciaaos escoamentos. O desenvolvimento do regime de golfadas em tubulaçõeshorizontais se dá a partir do escoamento estratificado em decorrência de doisfatores: do crescimento natural de pequenas perturbações (por um mecanismo deinstabilidade do tipo Kelvin-Helmholtz) ou devido à acumulação de líquidocausada por mudanças de inclinação no perfil do duto. O presente trabalhoconsiste da simulação numérica do surgimento das golfadas em ambas assituações descritas acima, assim como do subseqüente desenvolvimento doescoamento neste padrão para um regime estatisticamente permanente. Aprevisão do escoamento é obtida utilizando-se uma formulação unidimensionalbaseada no Modelo de Dois Fluidos. Parâmetros médios das golfadas(comprimento, velocidade e freqüência) são comparados com estudos numéricose experimentais da literatura, obtendo-se uma concordância bastante satisfatória, especialmente dada a simplicidade de uma formulação unidimensional.

Because of the complexity of two-phase flow phenomena, two-phase flow codes rely heavily on empirical correlations. This approach has a number of serious shortcomings. Advances in parallel computing and continuing improvements in computer speed and memory have stimulated the development of numerical simulation tools that rely less on empirical correlations and more on fundamental physics. The objective of this work is to take advantage of developments in massively parallel computing, single-phase computational fluid dynamics of complex systems, and numerical methods for front capturing in two-phase flows to develop a computer code for direct numerical simulation of two-phase flow. This includes bubble/droplet transport, interface deformation and topology change, bubble/droplet interactions, interface mass, momentum and energy transfer.