CFD SIMULATION OF BUBBLY TWO-PHASE FLOW IN HORIZONTAL PIPES

MOHD SAIDI BIN WAN AHMAD

Submitted to the Faculty of Chemical & Natural Resources Engineering in partially fulfillment of the requirements for the degree of Bachelor of Chemical Engineering (Gas Technology)

Faculty of Chemical & Natural Resources Engineering
University Malaysia Pahang

APRIL 2009

I declare that this thesis entitled “CFD Simulation of Bubbly Two-Phase Flow in Horizontal Pipes” is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : ....

Name : Mohd Saidi Bin Wan Ahmad

Date :

Dedicated, in thankful appreciation for support,
encouragement and understanding
to my beloved family and friends.

AKNOWLEDGEMENT

First of all, I would like to express my heartily gratitude to my research supervisor, Associate Professor Zulkafli Bin Hassan for his guidance, advices, efforts, supervision and enthusiasm given throughout for the progress of this research.

In preparing this thesis, I was in contact with many people, lecturers, and training engineers. They have contributed towards my understanding and thoughts. Without their continued support and interest, this thesis would not have been the same as presented here.

I would like to express my sincere appreciation to my parents for their support to me all this year. Without them, I would not be able to complete this research. Besides that, I would like to thank my course mates and my friends especially my housemate E32 for their help, assistance and support and encouragement.

ABSTRACT

Modeling of two phase flow, particularly liquid-vapor flow under diabatic conditions inside a horizontal pipe using CFD simulation is difficult with the available two phase models in FLUENT due to continuously changing flow patterns. In the present analysis, CFD analysis of air-water bubbly two-phase flow inside a horizontal pipe of inner diameter, 38.1 mm and 2000 mm length has been modeled using the volume averaged multiphase flow equations. Liquid volumetric superficial velocities varied at constant 1.56 m/s and gas volumetric superficial velocities varied in the range from 0.15 to 0.8 m/s. The average gas volume fraction varied in the range from 4% to 16%. The predicted gas volume fraction and mean gas velocity are compared with the experimental data that has been run before. The model prediction shows better agreement with experimental data is obtained using k-ε model with constant bubble size (1 mm). The results indicate that the volume fraction has a maximum near the upper pipe wall, and the internal flow structure is varies with increasing superficial gas velocity. It was found that increasing the superficial gas velocity at fixed superficial liquid velocity would increase the local gas volume fraction. The simulation results were consistent with experimental observations. An interesting feature of the liquid velocity distribution is that it tends to form a fully developed turbulent pipe flow profile at the lower part of the pipe, whereas in the top of the pipe a different flow exists.

ABSTRAK

Pembangunan model aliran dua fasa, terutama cecair dan udara yang mengalir dalam keadaan diabatic di dalam satu paip mengufuk menggunakan simulasi CFD adalah sukar kerana dua fasa yang terdapat di dalam model-model di FLUENT selalunya berubah-ubah corak alirannya. Analisis CFD untuk dua fasa iaitu air dan udara yang dijalankan dalam satu paip mengufuk berdiameter, 38.1 mm dan 2000 mm panjang telah dimodelkan menggunakan persamaan jumlah purata aliran yang berbilang fasa. Kelajuan air telah ditetapkan pada 1.56 m/s manakala kelajuan udara pada julat antara 0.15 hingga 0.80 m/s. Purata pecahan isipadu gas pula dianggarkan pada julat 4% hingga 1.6%. Nilai purata pecahan isipadu gas dan purata kelajuan gas akan dibandingkan dengan nilai eksperiment yang telah dijalankan sebelum menjalankan simulasi CFD. Bentuk model k-ε model yang disimulasi menunjukkan keputusan yang memuaskan apabila dibandingkan dengan data dari eksperiment di mana saiz gelembung udara ditetapkan pada 1mm. Keputusan menunjukkan pecahan isipadu gas mempunyai nilai yang maksimum pada bahagian atas paip dan struktur aliran dalaman adalah berbeza-beza apabila halaju udara dinaikan. Ini membuktikan bahawa apabila halaju udara dinaikan pada keadaan halaju air ditetapkan, pecahan isipadu gas pada bahagian atas paip turut meningkat. Keputusan dari simulasi CFD memberikan keputusan yang sama dengan keputusan dari eksperiment yang dijalankan.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CFD	–	Computational Fluid Dynamic
V_L	–	Liquid Velocity
V_G	–	Gas Velocity
$\sigma_k, \sigma_\varepsilon$	–	Constant in k-ε model
C_1, C_2	–	Constant is used k-ε model
g	–	Gravity acceleration (m/s^2)
k	–	Turbulent kinetic energy (m^2/s^2)
Re	–	Particle Reynold number
r	–	Radius of the pipe (m)
i.d	–	Internal diameter (m)
L	–	Length (m)

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Research Background

In the research of two-phase flow, not only are the flow characteristic and the performance of heat and mass transfer impacted by the difference of flow regime, but also the accurate measurement of two-phase flow parameters is. Therefore, the identification and prediction of flow regime are important to the design and operation of the engineering equipments.

1.1.1 Two Phase Flow

The flow conditions of two-phase flow always appear in industrial fields, such as dynamic force, chemical industry, nuclear power, refrigeration, petroleum, metallurgy etc. The research on two-phase flow becomes more and more important in national economy and human living [1]. Due to the complexity of multiphase flow behavior, it presents a great challenge to the study of the flow mechanisms and the measurement of multiphase flow. The accurate measurement of two-phase flow parameters has always been a key issue and tough work for decades. Although some achievements have been obtained, the solution is not yet satisfactory so far.

Researchers have made a large study in this area. Simultaneous flow of two or more immiscible phases is termed as multiphase flow. The common class of multiphase flow is he two-phase flow such as gas–liquid, gas–solid, liquid–liquid and solid–liquid flows. Gas–liquid flow is complex because of the existence of deformable interfaces and the fact that one of the phases is compressible. A wide range of interfacial configurations is possible in such two-phase flow [2].

The studies in two-phase flow through pipes have been conducted for the past 60 years. The first detailed study on two-phase flow was carried out by Lockhart and Martinelli in 1949. Design of pipeline for the simultaneous flow of oil and gas was discussed by Baker [3] and Hoogendoorn [4] has studied the gas–liquid flow in horizontal pipes. The two-phase slug flow in horizontal and inclined tubes was discussed by Vermeulen and Ryan [5]. Beretta et al. [6] has studied pressure drop for horizontal oil–water flow in small diameter tubes. In another study Awwad et al. [7] analyzed flow patterns and pressure drop with air–water in horizontal helicoidal pipes. A comparison of existing theories on two-phase flow was analyzed by Kordyban [8]. Pressure gradient due to friction for the two-phase mixtures in smooth tubes and channels was studied by Chisolm [9]. They are the successors in the field of two-phase flow and the studies were concentrated on developing the flow pattern model for horizontal, inclined and vertical flow in pipes.

1.1.2 History of Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a computational technology that enables us to study the dynamics of things that flow. Using CFD, we can build a computational model that represents a system or device that we want to study. Then we can apply the fluid flow physics and chemistry to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena. Therefore, CFD is a sophisticated computationally-based design and analysis technique.

CFD software gives us the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. Using CFD software, we can build a 'virtual prototype' of the system or device that we wish to analyze and then apply real-world physics and chemistry to the model, and the software will provide us with images and data, which predict the performance of that design. Computational fluid dynamics (CFD) presents an excellent tool for accomplishing such an objective because of its ability to simulate chemically reactive flows, heat and mass transfer, as well as mixing and related phenomena involving turbulence [10].

The fundamental basis of any CFD problem is the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, these equations can be linearized to yield the linearized potential equations. Historically, methods were first developed to solve the Linearized Potential equations. Two-dimensional methods, using conformal transformations of the flow about a cylinder to the flow about an airfoil were developed in the 1930s. The computer power available paced development of three-dimensional methods. The first paper on a practical three-dimensional method to solve the linearized potential equations was published by John Hess and A.M.O. Smith of Douglas Aircraft in 1966. This method discretized the surface of the geometry with panels, giving rise to this class of programs being called Panel Methods [11].

The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for inviscid, and Navier-Stokes equations for viscous flow). In addition, such a mesh can be either irregular (for instance consisting of triangles in 2D, or pyramidal solids in 3D) or regular; the distinguishing characteristic of the former is that each cell must be stored separately in memory. Where shocks or discontinuities are present, high resolution schemes such as Total Variation Diminishing (TVD), Flux Corrected Transport (FCT), Essentially NonOscillatory (ENO), or MUSCL schemes are needed to avoid spurious oscillations (Gibbs phenomenon) in the solution.

It is possible to directly solve the Navier-Stokes equations for laminar flows and for turbulent flows when all of the relevant length scales can be resolved by the grid (a Direct numerical simulation). In general however, the range of length scales appropriate to the problem is larger than even today's massively parallel computers can model. In these cases, turbulent flow simulations require the introduction of a turbulence model. Large eddy simulations (LES) and the Reynolds-averaged Navier-Stokes equations (RANS) formulation, with the k-ε model or the Reynolds stress model, are two techniques for dealing with these scales.

In many instances, other equations are solved simultaneously with the Navier-Stokes equations. These other equations can include those describing species concentration (mass transfer), chemical reactions, heat transfer, etc. More advanced codes allow the simulation of more complex cases involving multi-phase flows (e.g. liquid/gas, solid/gas, liquid/solid), non-Newtonian fluids (such as blood), or chemically reacting flows (such as combustion).

1.1.3 History of Pipeline system

A pipeline system is a transportation network of pipes, valves, and other parts connected together to deliver gaseous or liquid products from a source (supplier) to a final destination. In terms of PGBs operation our pipelines are build under the Peninsular Gas Utilization (PGU) project, gas from offshore fields is landed at the Gas Processing Plant (GPP) in Kertih,Terengganu, where the gas is treated and processed. After the natural gas is processed, it is put into pipelines that transport the natural gas across the country. These lines are called transmission lines, and they cross all over Malaysia. They are usually large diameter pipelines and can be several hundred kilometers long for example the PGU Loop 1 transmission line from Kertih to Segamat, which is 270KM in distance. These lines operate at very high pressures, so large quantities of natural gas can be moved across the country. Pressure in transmission lines can vary from about 500 pounds per square inch (psi) up to 1400 psi or more. Besides meeting the domestic demand, gas is also being exported to Singapore since January 1992 for use at the Senoko power station [12].

1.2 Statement of the Problem

In designing the pipeline, the prediction of internal flow structure is as important as the prediction of pressure drop. The devices that operate under bubbly flow conditions have large interfacial areas for heat and mass transfer. In this experiment, we focus on the effect of bubbly in water flow. Horizontal flows have received less attention in the research than vertical flows. Experimental observations are also difficult in this case, as the migration of dispersed bubbles towards the top of the pipe, due to buoyancy, causes a highly non-symmetric volume fraction distribution in the pipe cross-section. The problems of characterizing internal flow structure of gas-liquid mixtures flow in horizontal pipes not always exhibit a fully developed equilibrium condition. Under certain conditions there may be periodic wavy flow in the axial directions. The expansion of the gas phase associated with the frictional pressure gradient also causes a continuous acceleration of the mixture, and, consequently, a continuous flow development in the axial direction.

1.3 Objectives

1. To develop hand on skill developing model using CFD simulation software
2. To investigate the internal flow structures of bubbly two-phase flow in horizontal pipe.
3. To validate the experimental work data with the CFD simulation data based on gas volume fraction, liquid velocity, gas velocity and turbulence field in horizontal two phase pipe flow.

1.4 Scopes of Research Work

1. To simulate the effect of liquid velocity and gas volume fraction due to internal flow structure of bubbly in horizontal pipe using experimental work and CFD simulation.
2. To figure out the flow patterns relevant to liquid flow rate through a horizontal pipe based on experimental work.
3. To analyst and comparison between both data from the CFD simulation and experimental work.

CHAPTER 2

LITERATURE REVIEW

2.1 Background of the Research

In designing of pipeline, the prediction of pressure drop is as important as the prediction of heat transfer coefficient. Modeling of two phase flow, particularly liquid – vapor flow under adiabatic conditions inside a horizontal pipe using CFD analysis is difficult with the available two phase models in FLUENT due to continuously changing flow patterns. The resulting two phase flow is more complicated physically than single phase flow. In addition to the usual inertia, viscous and pressure forces present in single phase flow, two phase flows are also affected by interfacial tension forces, the wetting characteristics of the liquid on the tube wall and the exchange of momentum between the liquid and vapor phases in the flow. Because of these effects, the morphology of two phase flow patterns varies for different geometries of channels or pipe and their orientations [13].

Basic one dimensional model of single component two phase flows is developed for the stratified flow where each phase is in contact with the channel and has a common interface. From the separated flow model, the frictional pressure drop for two phase, two component, isothermal flow in horizontal tubes was initially developed by Lockhart and Martinelli in 1944 [14] using the two phase multiplier. A later extension of their work to cover the accelerative component resulted into well known Martinelli-Nelson correlation for the prediction of pressure drop for forced circulation boiling and condensation. Later, the calculation methodology for two phase friction multiplier was developed by Thom [14], Baroczy [14] and Chisholm [14].

In the recent literature, Tribbe and Muller-Steinhagen [15] presented an extensive comparison of 35 two phase pressure drop predictive methods compared to a large database for the following fluid combinations: air-oil, cryogenics, steam-water, air-water and several refrigerants. They made a statistical comparison for this large database also segregating the data by fluid. They found that statistically the method of Muller-Steinhagen and Heck [16] gave the best and most reliable results. In the revised edition, Chapter 13 of Engineering Data book III of the Wolverine Tube Inc., [17] it is mentioned that overall, the Gronnerud and the Muller-Steinhagen and Heck methods to be equally the best, while the Friedel method was the third best in a comparison of seven leading predictive methods.

Kattan et al. [18] segregated the data by flow regimes using the flow pattern map and the authors found that predictive methods work differently with varying the flow regime, since the models are not able to capture completely the effects of the variations in flow structure. Recently, Moreno Quiben and Thome [19, 20] published a work in which they made a comprehensive study to run accurate experiments. Then using a new flow pattern map by Wojtan et al. [21], they built a flow pattern based model for predicting pressure drops. However, there is not much data reported in the literature on modeling and analysis of two phase flow using CFD software.

2.2 Multiphase Flow

Multiphase flows are defined as flow of a heterogeneous mixture of multiple fluids or phases, where the fluid or solid particles can be identified as macroscopic structures. So the fluids in a multiphase flow are not homogeneously mixed at a molecular level, but macroscopic regions of the one or the other fluid or phase can be observed, e.g. solid particles, droplets, bubbles, slugs, liquid films or ligaments, etc. Typical examples of such kind of multiphase flows are bubbly flows, sprays, gas- or liquid-solid flows (e.g. in sand blasting, abrasive-jet or water-jet cutting applications), but also stratified flows where fluids are separated by a free surface like in annular and slug flow regime of gas-liquid two-phase flows in pipes and channels. Due to the presence of multiphase flows in many industrial applications there numerical prediction by means of state-of-the-art CFD is of common interest for a multitude of industrial branches. So multiphase flows play an important role e.g. in power generation (from fossil fuels, in water and nuclear power plants), in nuclear and chemical reactor safety technology, in food processing industry, in chemical and mechanical process technology as well as in processes in the automobile, aeronautic and space industry.

The main difficulty in the physico-mathematical description of multiphase flows arises from the fact, that for most multiphase flows the flow morphology and thereby the shape and interfacial area of the phase interface separating the two or more phases of the multiphase mixture is priory unknown. The numerical prediction of a multiphase flow can become further complicated, if phase change processes or chemical reactions are part of the application, leading to mass, momentum and heat transfer between phases or fluids. Physical modeling is required, since not all length scales of the flow morphology and not all microscopic processes at the phase interface can be described or resolved in full detail with there spatial and temporal distribution.

In order to reduce the complexity of the problem, the present paper concentrates on the development and application of contemporary CFD models for gas-liquid two-phase flows, also the main concepts and ideas are applicable to gas-solid and liquid-