MODELLING OF HYDRODYNAMICS IN AIRLIFT REACTOR

NUR HAZIEQAH KAHIL

Thesis submitted in partial fulfilment of the requirements
for the award of the degree of
Bachelor of Chemical Engineering (Gas Technology)

Faculty of Chemical & Natural Resources Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2013

ABSTRACT

Airlift reactors are widely used in chemical, petrochemical and biochemical industries. This type of reactor is much more productive in terms of specific power demands and commercial scale-effectiveness. Concept of hydrodynamics in airlift reactors is influenced by these two parameters; axial gas velocity and axial liquid velocity. It is important to understand the concept as the parameters affected to all aspects in performance of airlift reactors. Study of hydrodynamics has been started since decades ago, but experimental study is not a first choice for the research due to an expensive setup to develop. Experimental study using Computer Automated Radioactive Particle Tracking (CARPT) and Laser Doppler Anemometry (LDA) are examples of experimental study on airlift reactors. Basically, these experimental study take lots of times to scale-up the prototype and costly. Thus, Computational Fluid Dynamics (CFD) is used as an alternative method as the cost for the study is much cheaper and even better in performing the result. GAMBIT 2.2 and FLUENT 6.3 are used to evaluate the performance of airlift reactors. Several phases involved in this study by suing two Eulerian models; mixture k-ε model and two phase k-ε model. The experimental literature from Van Baten et al. (2003) is chosen as the validation data for CFD simulation. The CFD predicts the axial component of gas velocity and axial component of liquid velocity fairly well, although the results seem to suggest that further improvement need to be studied. It is clear from the modelling exercise performed in this work that CFD is a great method for modelling the performance of airlift reactor.

ABSTRAK

Reaktor angkutan udara digunakan secara meluas dalam industry kimia, petrokimia dan bio-kimia. Reaktor jenis ini lebih produktif dalam permintaan kuasa khusus dan keberkesanannya secara skala komersial. Konsep hidrodinamika di dalam reaktor ini dipengaruhi oleh dua parameter seperti kelajuan gas dan kelajuan cecair. Ini adalah sangat penting untuk memahami konsep parameter tersebut yang mempengaruhi prestasi reaktor angkutan udara. Kajian mengenai konsep hidrodinamika telah dilakukan beberapa dekad yang lepas, namun kajian secara eksperimentasi bukanlah pilihan utama kerana kos yang agak mahal untuk membina prototaip tersebut. Kajian secara eksperimen seperti Computer Automated Radioactive Particle Tracking (CARPT) dan Laser Doppler Anemometry (LDA) adalah teknik yang selalu digunakan dalam kaedah eksperimen. Secara amnya, kedua-dua teknik ini memakan masa untuk disiapkan dan kosnya yang mahal. Oleh itu, pengkomputeran bendalir dinamik (CFD) digunakan sebagai kaedah alternatif yang lebih mudah, jimat dan menjanjikan prestasi yang lebih baik. GAMBIT 2.2 dan FLUENT 6.3 adalah dua perisian yang digunakan untuk mengkaji prestasi reaktor angkutan udara. Pelbagai fasa simulasi telah digunakan dengan menggunakan konsep Eulerian iaitu model campuran k-ε dan model dua fasa k-ε . Kajian secara eksperimentasi oleh Van Baten et al. (2003) dijadikan rujukan untuk memastikan kesahihan data yang diperoleh daripada simulasi CFD. Data kelajuan gas dan kelajuan cecair yang diramalkan menggunakan simulasi menunjukkan hasil yang memberangsangkan walaupun keputusan tersebut memerlukan kajian yang lebih mendalam dan terperinci. Justeru, jelas di sini bahawa simulasi CFD mampu menghasilkan data yang baik untuk mengkaji prestasi reaktor angkutan udara.

TABLE OF CONTENTS

SUPERVISOR'S DECLARATION IV
STUDENT'S DECLARATION V
Dedication VI
ACKNOWLEDGEMENT VII
ABSTRACT VIII
ABSTRAK IX
TABLE OF CONTENTS X
LIST OF FIGURES XII
LIST OF TABLES XIII
LIST OF ABBREVIATIONS XIV
LIST OF ABBREVIATIONS XV
1 INTRODUCTION 1
1.1 Motivation and statement of problem 1
1.2 Objectives 2
1.3 Scope of this research 2
1.4 Main contribution of this work 2
1.5 Organisation of this thesis 2
2 LITERATURE REVIEW 4
2.1 Overview 4
2.2 Introduction 4
2.3 Previous work on hydrodynamic in airlift reactor 5
2.3.1 Computer Automated Radioactive Particle Tracking (CARPT) 5
2.3.2 Laser Doppler Anemometry (LDA) 6
2.3.3 Particle Image Velocimetry (PIV) 7
2.4 Application of airlift reactors 7
2.5 Hydrodynamics of Airlift Reactor 8
2.5.1 Study of Axial Component of Gas Velocity 8
2.5.2 Study of Axial Component of Liquid Velocity 9
2.6 CFD Modelling and Operation of Airlift Reactor 9
2.7 Summary 10
3 MATERIALS AND METHODS 11
3.1 Overview 11
3.2 Introduction 11
3.2.1 Identifying Physical Problem 11
3.2.2 Pre-processing / Meshing 11
3.2.3 Iteration / Numerical Method Setup 13
3.2.4 Post-processing in FLUENT 13
3.2.5 Verification of CFD Model 13
3.3 CFD Modelling of Two Phase Flow 14
3.4 Turbulence Modelling 15
3.4.1 Dispersed k-ε model 16
3.5 Airlift Reactor Dimension and Modelling Strategy 16
3.6 Summary 17

4 EXTRACTION OF HYDRODYNAMICS IN AIRLIFT REACTOR....................18
4.1 Overview 18
4.2 Assessment and Prediction of Axial Component of Gas Velocity in Airlift Reactor....................18
4.3 Assessment and Prediction of Axial Liquid Velocity Profile in Airlift Reactor....................22
4.4 Summary....................25
5 CONCLUSION 26
5.1 Conclusion....................26
5.2 Future work....................26
REFRENCES....................27
APPENDICES....................29

LIST OF FIGURES

Figure 2-1: Schematic of an airlift reactor..4
Figure 2-3: a) Aeration tank with pump system b) Aeration tank with airlift system.......7
Figure 3-1: Surface mesh of bubble column with it dimension.
(a)side view and (b)inlet view……………………………………………………11
Figure 4-1: Comparison between simulated and experimental profile of axial gas velocity:
a) U_G = 0.04 m/s..18
b) U_G = 0.06 m/s..18
c) U_G = 0.08 m/s..19
d) Contours vector of gas velocity...19
Figure 4-2: Contours of volume fraction – gas velocity..20
Figure 4-3: Comparison between simulated and experimental profile of axial gas velocity:
a) U_G = 0.04 m/s..21
b) U_G = 0.06 m/s..22
c) U_G = 0.08 m/s..22
d) Contours vector of liquid velocity…………………………………………………23

LIST OF TABLES

Table 3-1:Properties used in the CFD simulations……………………………………..15

LIST OF ABBREVIATIONS

C_D	drag coefficient
C_L	lift coefficient
$C_{\varepsilon 1}$	constant for eqs. 12
d_b	Sauter mean bubble diameter
d_b	bubble size
$\vec{F}_{\mathrm{lg}}$	interaction force mainly due to drag
$\vec{F}_{lift}$	lift force
$\vec{F}_{vm}$	virtual mass force
g	gravity acceleration
G_k	turbulent production term
k	turbulent kinetic energy
P	pressure
v_{sg}	superficial gas velocity
t	time
u, v	velocity components
u_t	turbulent viscosity

Greek

α	void fraction
ε	turbulent dissipation rate
ρ	density
σ_ε	constant for eqs. 8
σ_k	constant for eqs. 9
$\Pi_{k,l}$	characteristic turbulent kinetic energy for secondary phase
$\Pi_{\varepsilon,l}$	characteristic turbulent dissipation rate for secondary phase
$\bar{\bar{\tau}}_l$	liquid phase stress-strain tensor
μ_l	liquid viscosity
μ_m	turbulent viscosity

Subscripts

b	bubble
g	gas
l	liquid
m	mixture
i	mixture entity of *i* phase

LIST OF ABBREVIATIONS

CARPT	Computer-Automated Radioactive Particle Tracking
CFD	Computational Fluid Dynamics
LDA	Laser Doppler Anemometry
LDV	Laser Doppler Velocimetry
PIV	Particle Image Velocimetry
PMT	Photomultiplier tube

1 INTRODUCTION

1.1 Motivation and statement of problem

Airlift reactors (ALR) are finding increasing application in industry especially in chemical industry, biochemical fermentation and biological wastewater treatment processes (Van Baten et al., 2003). These reactors are reaction vessels divided into two main sections which contain the riser, where the gas is usually injected, and the downcomer. Since there is a vertical baffle or a concentric draft tube for dividing the sections, only the riser zone is generally sparged with the gas. As a result of the different gas holdup in the riser and in the downcomer, the bulk density of the fluid in these zones is different and liquid circulation is induced. This difference causes the fluid in the riser to flow upward and fluid in the down-comer zone to move downwards to create a well defined circulatory flow chanelled through the riser and the down-comer (Moraveji, 2012).

For design a good airlift reactor, it is necessary to estimate gas hold-up and liquid circulation velocity of airlift reactors (Lin and Chen, 2005). Several literature studies also focused on these two hydrodynamic parameters (Van Baten et al., 2003), (Moraveji, 2012), (Gimbun, 2009), (Gajbhiye et al., 2012). These two parameters clearly influenced all aspects on the mixing and mass transfer performance of the airlift systems. Airlift reactors are basically modified bubble column reactors where the riser section itself can be regarded as a bubble column. The fluid dynamic condition for the airlift reactors has been altered to be different from that of the bubble column (Lin and Chen, 2005). An alteration has been made to airlift reactors for increasing the performance of other reactors to more productive in terms of specific power demands and commercial-scale effectiveness (Gajbhiye et al., 2012).

For a few decades, studies in hydrodynamics for airlift reactors using computational fluid dynamics (CFD) simulation have been made. Those researches have been developed to achieve better design of airlift reactors for the industry. Most of the studies are focused on two fluid models, which assume the gas and liquid phases as an interpenetrating media. Usually the Eulerian two-fluid model is applied to solve two phase problem and the dispersed k-ε model is used for turbulence modeling.

1.2 Objectives

The following are the objectives of this research:

- To study hydrodynamics of gas-liquid multiphase flow in an airlift reactor via 3D Computational Fluid Dynamics simulation.

1.3 Scope of this research

The following are the scope of this research:

i) Modelling for airlift reactor by comparing with the experimental data from previous study (Van Baten et al., 2003).

ii) Experimental analysis of hydrodynamics and performance of liquid velocity in airlift reactors.

iii) Experimental analysis of hydrodynamics and performance of gas velocity in airlift reactors.

1.4 Main contribution of this work

The following are the contributions:

Understanding the operation and hydrodynamics are important for both design and scale-up purpose. A high cost and plenty time are needed to develop a prototype for experimental testing. Thus, this research has been carried out to give a faster solution as well as much more economical.

CFD simulation provide a similar result as the experimental result or even better for multiphase flow depends on the selection of turbulence model for gas-liquid modelling as well as contribute a new development of an advanced technology for the research.

1.5 Organisation of this thesis

The structure of the reminder of the thesis is outlined as follow:

Chapter 2 provides a description of the applications and general design features of airlift reactors. A general description on the flow characteristics of the system, as well as the dimensionless groups and correlations to account for the flow phenomena are presented. This chapter also provides a brief discussion of the advanced experimental techniques available for airlift reactors, mentioning their applications and limitations for bubbly flow analysis. A summary of the previous experimental work on airlift reactor is also presented. A brief discussion on the scale-up methods for airlift reactors is also provided.

Chapter 3 gives a review of the CFD approach applied for airlift reactors modelling of multi-phase flow including the turbulence modelling and solution procedures. Mathematical modelling which describe the fluid flow phenomena involved in an airlift reactor and the modelling strategy are presented.

Chapter 4 is a comprehensive comparison of the simulation results and the experimental data on gas hold-up profile, turbulent kinetic energy and axial gas-liquid velocity with detailed discussion.

Chapter 5 draws together a summary of the thesis and outlines the future work which might be derived from the model developed in this work.

2 LITERATURE REVIEW

2.1 Overview

This chapter presents the current development on CFD modelling of hydrodynamics in airlift reactors. A crucial review on simulation approach that probably affects the performance of airlift reactors such as axial liquid velocity, gas holdup profile and axial gas velocity as the main interest in this study. Furthermore, a brief summary about the experimental measurement technique for predicting the hydrodynamics of airlift reactors was also discussed.

2.2 Introduction

Airlift reactors are an important class of multiphase reactors that are widely used in the chemical industry, biotechnology based production of diverse products and biological treatments. It consists of a liquid pool divided into two distinct zones which is usually sparged by a gas. The difference of bulk densities that exist in a fluid region also known as a gas holdup causes a circulation of the fluid in the reactor by a gas-lift reaction. In the reactor, the part which consists the gas-liquid upflow is the riser while the downflowing fluid region is known as the downcomer (Figure 2.1).

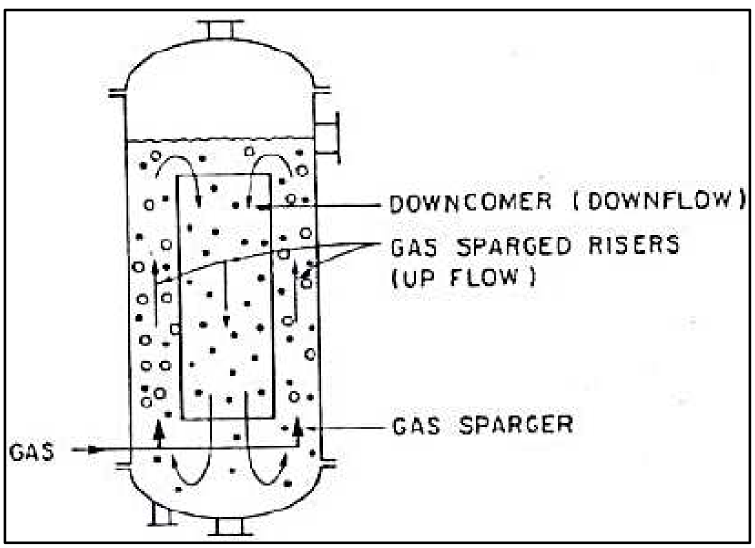

Figure 2-1: Schematic of an airlift reactor

Theoretically, airlift reactors may be employed for any gas-liquid or gas-slurry contacting process. The mixing fluid and mass transfer of gas-liquid in an airlift reactor could be achieved using a compression of gas which mostly is air. Besides, there is no mechanical agitation used inside the reactor. Hence, it can be simplified to reduce the power and maintenance costs (Moraveji, 2012). The application of the reactors depends on the ability to achieve rates of momentum, heat and mass transfer at acceptable capital and operating costs. The technical and the economic feasibility of using airlift devices has been conclusively established for a number of processes and widely increasing use in aerobic fermentations, wastewater treatment and other similar operations.

Modelling of hydrodynamics in an airlift reactor has been carried out in this research for three reasons: first to study the axial gas velocity and axial liquid velocity behaviour in airlift reactors; second to evaluate the suitability of the modelling approach and third, to validate the modelling methods by comparing the simulation results with experimental data from previous studies. In this study, a cylindrical airlift reactor was simulated, filled with tap water and the bubble size is assumed constant. The geometry of the airlift reactor studied was similar to the experimentally and simulated numerically by Van Baten et al. (2003).

2.3 Previous work on hydrodynamic in airlift reactor

2.3.1 Computer Automated Radioactive Particle Tracking (CARPT)

Computer Automated Radioactive Particle Tracking (CARPT) was first used by Devanathan (1991) which he applied radioactive particle tracking in bubble columns to study the motion of the liquid phase. A single radioactive article, Scandium, Sc^{46} is used due to its characteristics neutrally buoyant when the liquid phase being tracked. In the experiment, the particle is moved in the liquid phase of column tracking which is agitated by the gas passage. Then, the position of the particle is determined by an array of scintillation detectors that configure the emission of γ radiation by the particle. Emission of γ radiation with different intensities recorded by the detectors are decreased exponentially with increasing distance between the particle and the detector. At this point, calibration is performed using CARPT experiment for estimating the position of particle from the radiation intensities by placing the particle at various locations and monitoring the radiation recorded by each detector. By using the information required, calibration curves are established which relate the intensity to the distance between the detector and the particle. Thus, a set of instantaneous position data can be obtained that gives the positions of the particle at successive sampling instants by using a weighted regression scheme. Luo et al.(2008) had studied the local characteristics of hydrodynamics in draft tube airlift bioreactor by using CARPT technique. This study is focused on multiphase flow; macro mixing and turbulence intensity.

2.3.2 Laser Doppler Anemometry (LDA)

Laser Doppler anemometry (LDA) is an optical technique ideal for non-intrusive 1D, 2D and 3D point measurement of velocity and turbulence in both free flows and internal flows. It also known as Laser Doppler Velocimetry (LDV). The special feature of this technique is no calibration required by means the velocity value can be from 0 to supersonic and the components for the velocity can be run simultaneously. Besides, the measurement distances are available from centimetres to meters and the reversal of flow can be measured. Furthermore, due to its non-intrusive measurement principle, it is very suitable for applications with reversing flow, chemically reacting or high-temperature media and rotating machinery, where physical sensors are difficult or impossible to use. The basic configurations for LDA consists of a continuous wave laser, transmitting optics with a beam splitter and a focusing lens, receiving optics; a focusing lens, an interference filter and a photodetector with a signal conditioner and a signal processor. Basic concept for this technique is when a flow is seeded with small, neutrally buoyant particles which scatter light. A laser light will illuminated the particles to be detected by a photomultiplier tube (PMT) that generates a current in proportion to absorbed photon energy, and then amplifies the current. The difference between the incident and scattered light frequencies is called Doppler shift. Experimental study on local hydrodynamic behaviour of three-phase external-loop airlift reactor by Lin et al.(2004) which focused on the liquid velocity, bubble rise velocity, and gas holdup, were using a fiber-optic probe and laser Doppler anemometry techniques.

2.3.3 Particle Image Velocimetry (PIV)

Particle image velocimetry is an optical method of flow visualization to obtain instantaneous velocity measurements and related properties in fluids. Due to its unique features of being non-intrusive and whole-field this is simultaneously measuring the velocities of fluids at many points without any disturbance. The technique involves by seeding the fluid with tracer particles and illuminating the region under investigation. Two images measure shortly for its whole velocity fields and calculating the distance individual particles travelled within time. The PIV works when a flow is illuminated in the target area with a light sheet. Then, a camera lens images the target area onto the sensor array of a digital camera where the camera is able to capture each light pulse in separate image frames. Once a sequence of two light pulses is recorded, the images are divided into small subsections which called interrogation aeas (IA). This interrogation areas from each image frame, are cross-correlated with other, pixel by pixel. A signal peak is produced by the correlation which identify the common particle displacement. An accurate measure of the displacement and the velocity are achieved with sub-pixel interpolation. This method has been used in measuring the typical two phase flow of various work conditions in Membrane Micropore Aeration Bioreactor (MMAB) by Dong Liu et al. (2008).

2.4 Application of airlift reactors

Airlift reactors are widely used in industry due to its advantages. Since one of the recognized characteristics of airlift reactors is the potential for scaling up and the relatively low power consumption for agitation and oxygenation, it is only natural that many processes related to wastewater treatment use this type of reactors.

Molly Precast is one of the companies that applied the theory of an airlift to wastewater treatment plant. Before applying theory of airlift to their treatment, they used pump for aeration, to pump effluent between the chambers and to discharge the clarified water.

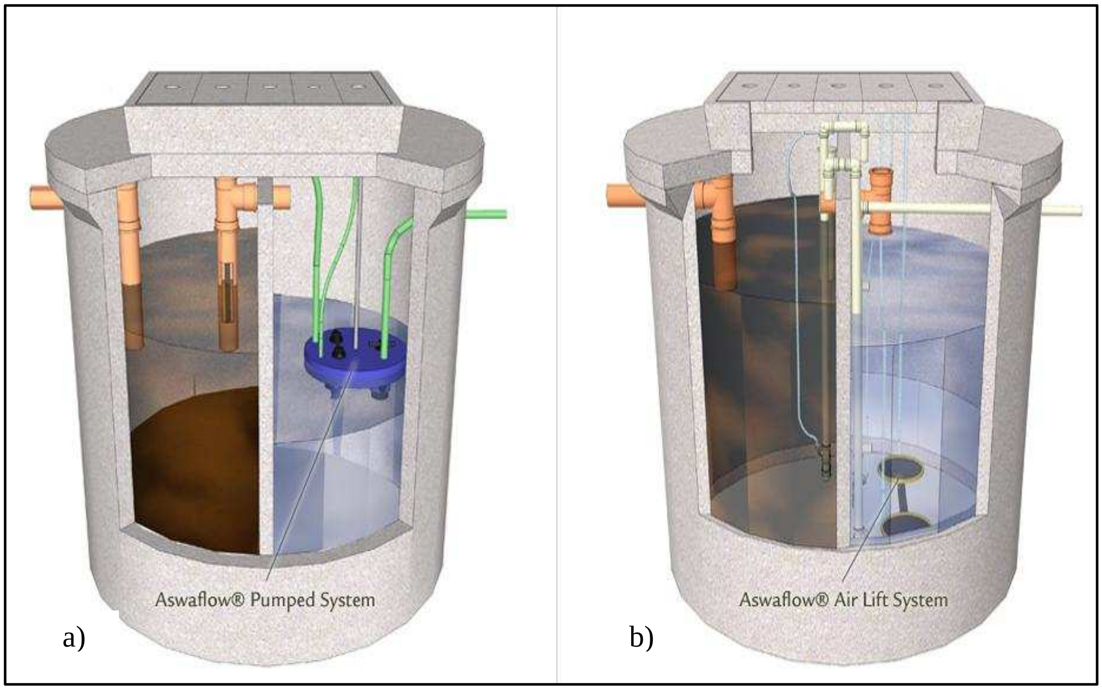

Figure 2-3: a) Aeration tank with pump system b) Aeration tank with airlift system (Source: *Molly Precast, 2013*)

Based from the figure above, both system have own characteristics and can be used for wastewater treatment. Pumped system is still relevant to be used for wastewater treatment but a system with an airlift technology performed much better where there is no pump and mechanical aerator involved. Besides, the airlift system much preferable since there is no electricity needed as well as reducing the costs.

2.5 Hydrodynamics of Airlift Reactor

2.5.1 Study of Axial Component of Gas Velocity

One of the most important hydrodynamic parameters involved in the design development, scale-up and troubleshooting of multiphase system is axial gas velocity. Gas hold-up is a dimensionless key parameter for design purposes that characterizes transport phenomena of airlift systems. It can be define as the volume fraction of gas phase occupied by the gas bubbles. Many studies examine gas holdup because it plays an important role in design and analysis of airlift system. As reported by Abashar et al. (1998), the important aspect in modelling the hydrodynamics of airlift reactors is the relationship between the dependent variable of gas holdup and liquid circulation rate and the independent variables of superficial gas velocity, the physical properties of the fluids and reactor geometry. The knowledge of gas hold-up in airlift reactors is important because has significant for the design and operation of transport systems and has a large application in industrial processes.

2.5.2 Study of Axial Component of Liquid Velocity

The liquid flow and mixing behaviour in airlift reactor can be described as the liquid recirculation velocity profile. The liquid circulation velocity affects the mixing and mass transfer rate in airlift reactors, since the gas holdup influences the gas-liquid mass transfer efficiency and the liquid circulation velocity, the study of gas holdup and axial liquid velocity is related. Due to the relationship between gas holdup and liquid circulation velocity, extensive studies on these two parameters had shown that the gas holdup and liquid circulation velocity increased as the aeration rate increased (Chang, 1994). The simulation can be used for prediction of the axial liquid velocity profile over arrange of conditions, which should help to improve the connection between these two parameters.

2.6 CFD Modelling and Operation of Airlift Reactor

CFD is one of the branches of fluid mechanics (Patel, 2010). It can predict the fluid flow, heat and mass transfer, chemical reactions and any related conditions by solving the mathematical equations by using numerical methods and algorithm.

For a few decades, many studies have established the potential of CFD for describing the hydrodynamics of airlift reactors (Van Baten et al, 2003). CFD is used to achieve better control and reliability due to the spectacular progress in digital computing (Moraveji, 2012).

Many studied have been done related with airlift reactors modelling and the simulations have been carried out for the predictions of flow pattern in the reactor using 1D, 2D and 3D models (Gimbun, 2009). Gas and liquid phase are two media that have been used widely for this research. This is equivalent to Eulerian approach where it is more reputed and applicable (Moraveji, 2012). However, most of those or recently studies still cannot predict correctly the information regarding airlift reactors operation behaviour as same as experimental results because there are many aspects need to be considered in the model, especially the multiphase fluid dynamics modelling and the interfacial exchange model. Therefore, the aim of this work is to develop a 3D CFD model to study the influence on gas hold-up profile and the liquid axial velocity in airlift reactors. The CFD prediction was compared to the experimental data from Van Baten et al. (2003).

2.7 Summary

Although many experimental methods are available currently, but each technique can only provide certain measurement (limited). Furthermore, experimental technique is highly investing in instrumentation and lots of times need to build a prototype. An advanced technology as CFD simulation is much preferable to study on airlift reactors system with a cheaper and faster solution. CFD are capable in simulating the gas hold-up profile, axial liquid velocity and turbulent kinetic energy of airlift reactors in multiphase flow regime. Many published works recommend CFD simulation to simulate gas liquid flow in three dimensional.

3 MATERIALS AND METHODS

3.1 *Overview*

This paper presents a computational fluid dynamics (CFD) of the gas-liquid flow in airlift reactors. Multiphase simulations were performed using an Eulerian-Eulerian two-fluid model. The CFD predictions were compared to the experimental measurement adopted from literature. The CFD predicts the turbulent kinetic energy, gas hold-up and the gas-liquid axial velocity well, although the results seem to suggest that further improvement to both the interfacial force model and two-fluid modeling approaches is necessary. It is clear from the modeling exercise performed in this work that CFD is a suitable method for modeling the performance of airlift reactors. Furthermore, the CFD method is certainly less expensive than the experimental characterization studies.

3.2 *Introduction*

In order to carry out the modelling, there are several steps that need to be accomplished to achieve a satisfactory result. The steps are as below:

a) Identify the physical problem
b) Pre-processing / Meshing
c) Iteration
d) Post-processing in FLUENT
e) Verification

3.2.1 Identifying Physical Problem

This is the first step on how to define the CFD simulating modelling goals. At this stage, physical model or the dimension of the model must be decided to be included in the analysis for design purposes.

3.2.2 Pre-processing / Meshing

The representing comparisons for multiphase stream were analyzed where the fluid flow domans are part into more diminutive subdomains to made up of geometric shape like hexahedral and tetrahedral in 3d. Then, the governing equation are discretized and solved inside each of these subdomains. Commonly, one of three routines is utilized to explain the approximate form of the system of equations: limited finite volumes, finite elements, or finite differences. The subdomains are regularly called elements or cells, and the gathering of all elements or cells is known as a mesh or grid. The procedure of getting a proper mesh or grid is called mesh generation or grid generation. Particular software programs have been created for these purposes and the triumph of a modeling is hinging upon expertise in utilizing this software. Gambit 2.4 programming was utilized to work the airlift reactor model as demonstrated in Figure 3-1.

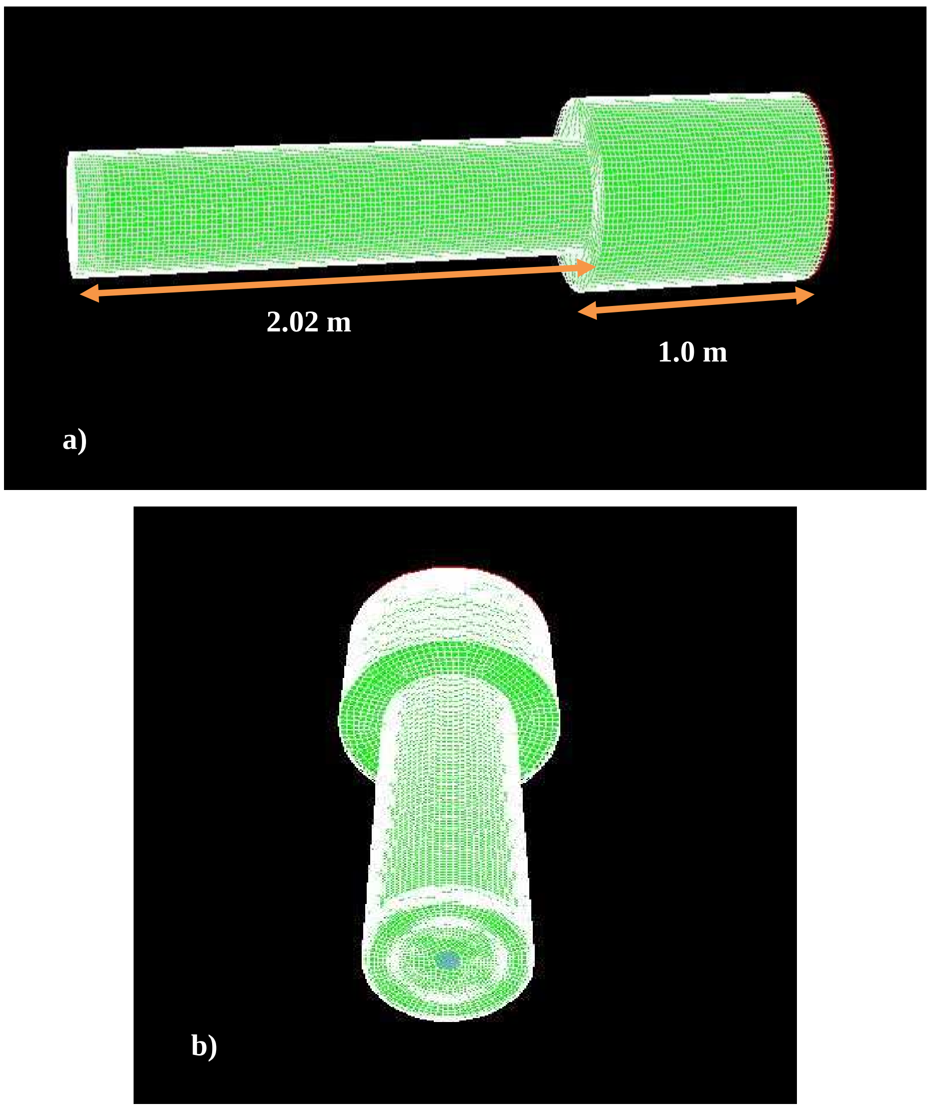

Figure 3-1: Surface mesh of airlift reactor with it dimension.

(a) side view and (b) inlet view

3.2.3 Iteration / Numerical Method Setup

At this stage, the boundary conditions specify the flow were setup on the boundaries of physical model. Thus, critical elements of FLUENT simulations are specified properly. The boundary types that were set for this simulation are as follow;

- Velocity inlet is assumed constant
- Outlet top set as pressure outlet
- Bottom inlet set as velocity inlet and multipoint sparge
- Bubble column wall set as standard wall function

After boundary condition has been set, result is numerically figured by the software. The discretized conservation equation is understood iteratively. Various iterations are typically needed to achieve a converged result. Convergence is reached when updates in result variables from one iteration to afterward are negligible and the general property conservation is achieved. The residue will furnish a mechanism to assist screen this trend. The precision of converge result is subject to appropriateness and accuracy of physical model, grid resolution and independence, and issue setup.

3.2.4 Post-processing in FLUENT

In this stage, the result is examined to review the result and remove the inconvenient data. The data examination is intended to guarantee the property conservation and correct physical behaviour. High residual may be attributable to only a few cells of poor quality.

3.2.5 Verification of CFD Model

The final stage if this study requires verification of simulation data with experimental data collected from literature (Van Baten, 2003). Comparison of simulation with experimental data will determine the accurateness of CFD prediction, where deviation less than 10% is considered satisfactory.

3.3 CFD Modelling of Two Phase Flow

The Eulerian-Eulerian approach is employed for gas-liquid airlift reactor simulation in this work, whereby the continuous and disperse phases are considered as interpenetrating media, identified by their local volume fractions. The volume fractions sum to unity and are governed by the following continuity equations:

$$\frac{\partial}{\partial t}(\alpha_l \rho_l) + \nabla \cdot (\alpha_l \rho_l \vec{u}_l) = 0 \tag{1}$$

where α_l is the liquid volume fraction, ρ_l is the density, and $\vec{u}_l$ is the velocity of the liquid phase. The mass transferred between phases is negligibly small and hence is not included in the right hand-side of eq. (1). A similar equation is solved for the volume fraction of the gas phase by replacing the subscript *l* with *g* for gas. The momentum balance for the liquid phase is:

$$\frac{\partial}{\partial t}(\alpha_l \rho_l \vec{u}_l) + \nabla \cdot (\alpha_l \rho_l \vec{u}_l \vec{u}_l) = -\alpha_l \nabla P + \nabla \cdot \bar{\bar{\tau}}_l + \vec{F}_{\mathrm{lg}} + \alpha_l \rho_l \vec{g} + \vec{F}_{lift,l} + \vec{F}_{vm,l} \tag{2}$$

where $\bar{\bar{\tau}}_l$ is the liquid phase stress-strain tensor, $\vec{F}_{lift,l}$ is a lift force, $\vec{g}$ is the acceleration due to gravity and $\vec{F}_{vm,l}$ is the virtual mass force. A similar equation is solved for the gas phase. $\vec{F}_{\mathrm{lg}}$ is the interaction force between phases, due to drag. Hence, $\vec{F}_{\mathrm{lg}}$ is represented by a simple interaction term for the drag force, given by:

$$\vec{F}_{\mathrm{lg}} = -\frac{3\alpha_g \alpha_l C_D |\vec{u}_g - \vec{u}_l|(\vec{u}_g - \vec{u}_l)}{4d_b} \tag{3}$$

where C_D is a drag coefficient and d_b is the Sauter mean bubble diameter.

The drag model employed has a significant effect on the flow field of the aerated flow, as it is related directly to the bubble terminal rise velocity. The drag model given as a function of the bubble Reynolds number, *Reb*, from Schiller and Naumann was employed in this work:

$$C_D = \frac{24}{Re_b}\left(1 + {Re_b}^{0.687}\right) \tag{4}$$

Lift forces act on a bubble due to the velocity gradients in the liquid phase and are said to be more significant for larger bubbles. The lift force acting on a gas phase in a liquid phase can be estimated from:

$$\vec{F}_{lift,g} = -C_L \rho_l \alpha_g \left(\vec{u}_l - \vec{u}_g\right) \times \left(\nabla \times \vec{u}_l\right) \tag{5}$$

where C_L is a lift coefficient has a value 0.5. A similar lift force is added to the right-hand side of the momentum equation for both phases ($\vec{F}_{lift,g} = -\vec{F}_{lift,l}$).

The virtual mass effect occurs when a gas phase accelerates relative to the liquid phase. The fluid surrounding the bubble is accelerating as a consequence of the bubble acceleration. This gives a rise to a force called a virtual mass which accounts for the losses of momentum of the accelerating bubble. The virtual mass force acting on bubbles is given by:

$$\vec{F}_{vm,g} = C_m \rho_l \alpha_g \left(\frac{d_l \vec{u}_l}{dt} - \frac{d_g \vec{u}_g}{dt}\right) \tag{6}$$

where C_m is the added mass coefficient has a value 0.5 for sphere. Similar with the lift force the virtual mass force is added to the right-hand side of the momentum equation for both phases ($\vec{F}_{vm,l} = -\vec{F}_{vm,g}$).

3.4 Turbulence Modelling

In turbulence modelling, there are three different options available for multiphase flow in FLUENT namely dispersed k-ε, mixture k-ε and two phase k-ε models (Gimbun, 2009). This paper used dispersed k-ε models due to easy to solve and calculated.