MODELLING OF OIL WATER PHASE INVERSION IN PIPELINE

WAN MOHAMED IRMAN BIN WAN MUSTAPHA

A thesis submitted in fulfilment of the
requirements for the award of the degree of
Bachelor of Chemical Engineering

Faculty of Chemical Engineering & Natural Resources
Universiti Malaysia PAHANG

May 2010

ABSTRACT

Dispersed oil-water two-phase pipe flow is frequently encountered in the petroleum and chemical processing industry. Conversion between these to types of dispersion is called phase inversion. The types of phase inversion present dependson several parameters such as the volume fraction of the phases, the viscosities of the liquids, the inter-facial tension and the turbulence in the flowing mixture. Predicting the phase inversion experimentally in a two-phase oil-water in pipeline is difficult due to the complexities measurements of the liquid hold-up and pressure drop at high velocities and volume fractions. Hence, a computational fluid dynamic (CFD) simulation using FLUENT 6.3 software was used to understand the flow behaviour. Both water-in-oil and oil-in-water dispersions were possible. In this research oil and water were used as a two-phase flow in pipeline with an internal diameter of 0.024 m and 9.7 m long. Data of the phase distribution profile and liquid hold-up was taken at 7.72 m from the inlet. The k-ε model was used to describe the turbulence in continuous phase. The numerical results from the simulation in terms of the phase distribution profiles and average in-situ hold-up are compared with experimental results by Soleimani (1999). Results that are presented and discussed shows acceptable agreement with the experimental data compared. From this work, it does appear that CFD simulation technique can be succesfully applied for the numerical simulation of liquid-liquid dispersed flow.

ABSTRAK

Minyak air-dua-peringkat terdispersi aliran paip sering dihadapi dalam industri petrokimia dan pemprosesan. Pertukaran antara jenis dispersi ini disebut Inversi fasa. Jenis Inversi fasa bergantung pada beberapa parameter seperti peratus isipadu fasa, kelikatan cecair, ketegangan antara-cecair dan arus dalam campuran mengalir. Ramalan Inversi fasa eksperimen di air-minyak dua-tahap dalam paip adalah kerana pengukuran kompleks sukar menahan cecair dan penurunan tekanan pada kelajuan tinggi dan fraksi kelantangan. Oleh kerana itu, dinamika fluida pengkomputeran (CFD) simulasi menggunakan perisian FLUENT 6.3 digunakan untuk memahami perilaku aliran. Kedua-dua air-dalam-minyak dan minyak-dalam-air dispersi itu mungkin. Dalam kajian ini minyak dan air digunakan sebagai aliran dua-fasa dalam paip dengan diameter dalaman 0.024 m dan 9,7 m panjang. Data profil pengedaran fasa dan cair terus-up diambil di 7,72 m dari inlet. Model k-ε digunakan untuk menggambarkan ombak di fasa kontinyu. Keputusan berangka dari simulasi dalam hal profil fasa pengedaran dan in-situ rata-rata terus up berbanding dengan hasil eksperimental dengan Soleimani (1999). Keputusan yang disajikan dan dibahas menunjukkan perjanjian diterima dengan data eksperimen berbanding. Dari karya ini, hal itu muncul bahawa teknik simulasi CFD boleh berjaya dilaksanakan untuk simulasi berangka cair-cair terdispersi mengalir.

TABLE OF CONTENTS

5 CONCLUSION AND RECOMMENDATION

LIST OF FIGURES

LIST OF SYMBOLS

$C\mu$ - Constant in standard κ-ε model
CD - Drag coefficient
D - Diffusion term
E_l - Liquid holdup
d_{32} - Energy change of reaction
f - Friction factor
G - Generation of turbulent kinetic energy, ms^{-3}
k - Turbulent kinetic energy, m^2s^{-2}
Re - Reynolds number
U - Velocity, ms^{-1}
V - Velocity, ms^{-1}
X - Martinelli parameter
Y - Distance from wall, m
x,y,z - Coordinate axis
α - Volume fraction
ε - Turbulent dissipation rate, m^2s^{-3}
μ - Viscosity, $kgm^{-1}s^{-1}$
ρ - Densitykgm^{-3}
φ - Turbulent kinetic energy or the dissipation rate of continuous phase
σ_{pr} - Prandtl number

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Motivation

Liquid-liquid dispersions ideally comprise of drops of one liquids dispersed in other liquid. Dispersions are widely applied in the petrochemical, food, chemical and pharmaceutical industries. In this research oil and water are used as a two-phase flow in pipeline. Both water-in-oil and oil-in-water dispersions were possible. Conversion between these two types of dispersion is called phase inversion. Phase inversion depends on several parameters such as the volume fraction of the two phases, the viscosities of the liquids, the inter-facial tension and the turbulence in the flowing mixture. The phase inversion phenomenon has been studied for many years (see for instance Becher, 2001). An often used model for phase inversion takes into account two opposite processes: break-up and coalescence of drops (Arashmid and Jeffreys, 1980; Nienow, 2004). Usually the break-up of drops is described by comparing the drop deformation due to turbulent pressure fluctuations with the deformation-restoring effect due to the inter-facial tension. The coalescence of drops is calculated by modeling the collision between drops. Phase inversion is then assumed to take place, when a certain criterion is satisfied. For instance, it is assumed that phase inversion occurs when the rate of breakup is not equal to the rate of coalescence, or that the viscosity of the mixture grows larger than a certain limiting value (Vaessen et al., 1996; Ioannou et al., 2004). In another model phase inversion is assumed to take place, when the inter-facial energy of the water-in-oil dispersion and the inter-facial energy of the oil-in-water dispersion are equal (Yeo et al., 2002; Brauner and Ullmann, 2002). For the calculation of the inter-facial energy it is necessary to know the drop size distribution and, therefore, again to model the break-up and coalescence processes. A study of dispersed oil-water turbulent flow in horizontal tube had been investigated numerically by Walvekar et al., (2009). The transient numerical simulations of two–phase dispersed flow in a pipe have been carried out using commercial CFD package FLUENT 6.2 in conjunction with multiphase model. In this study, drop breakup and coalescence is not considered and all the drops are assumed to of uniform size. Hence, the aim of the research is to study phase inversion in an oil-water flow through a pipeline. For that purpose, CFD package FLUENT 6.2 will be used to understand the oil-water phase inversion occurrence. Much attention was paid to the breakup of drops and droplet size distribution. The droplet size distribution will be calculated using population balance model or drag model for non-spherical droplet. In what follows, chapter 2 gives the objectives of the research, chapter 3 is the brief summary of the existing literature on this subject, chapter 4 is the method of CFD and chapter 5 is the conclusion and recommendation.

1.2 Description of Problem Statement

An understanding of the flow behavior within the pipeline is essential for equipment design, process scale-up, energy conservation and product quality control and can only be achieved by simulation and analysis of the multi-scale complex fluid dynamics involved in the mixing process. Experimental approaches to understand the flow behavior in pipeline are useful but the unsteady nature of turbulence flow, dispersed phase at which phase inversion occurs and the relative motion among fluid elements combine to make quantitative measurements and flow visualization both expensive and time consuming. Due to complexities in measurements of pressure drop and liquid hold-up at high phase fractions and velocities (Madhavan, 2005), dispersed liquid-liquid flows have not been studied in detail and very few numerical studies have been reported in the literature. Due to the complexities in measurements of system parameters especially at high mixture velocity and phase volume fractions, the present work focuses on computational fluid dynamics (CFD) simulation analysis.

1.3 Objective and Scope

This research aim to study numerically the two phase flow in pipeline. Oil-water mixture was considered as the two-phase flow with pipe of ID = 0.0024 m, 9.7 m long was used in the study. The following are the scopes of this study;

1) To investigate numerically the oil-water phase inversion in pipeline.
2) To study variation in water concentrations by taking into account the effect of drop break-up and coalescence.

1.4 Main Contribution

CFD simulations can result in major benefits for oil refining operation and equipment design by enabling engineers to understand and optimize processes. The use of computational physics in refinery has made it possible to analyze problems of greater complexity such as those involving reactions, multiphase flows, and complicated geometries, etc. In the present study, the CFD code FLUENT 6.2 is used to understand the liquid hold-up and its distribution for two phase (i.e., oil–water) mixture flowing through horizontal pipeline. The hold-up of the phase and its constituents is an important design parameter. The dispersed phase hold-up indicates the concentration the two phases in a mixture. The hold-up affects several transport processes, determines the global residence time of the dispersed phase and strongly influences the pressure drop in the system.

CHAPTER 2

LITERATURE REVIEW

2.1 Overview

Multiphase flow is the simultaneous flow of two or more phases in direct contact in a given system. It is important in many areas of chemical and process engineering and in the petroleum industry, e.g. in production wells and in subsea pipelines. The behavior of the flow will depend on the properties of the constituents, the flows and the geometry of the system.

There are four combinations of two-phase flows namely: gas-gas, gas-liquid, gas-solid, liquid-liquid, solid-solid and solid-liquid. Liquid-liquid flows, the subject of the present project are extremely important particularly in two-phase flow applications in horizontal pipes, for instance in the oil industry. In the oil industry, the dispersion of oil-in-water or vice versa usually appears in the oil well, to produce a fully oil in the well from offshore to onshore is one of the major problem for examples to investigate the physical of the pipe and the physical properties of the liquid that can affect the flow structure and production.

In liquid-liquid flow system, it is important to understand the nature of the interactions between the phases and to observe the ways in which the phases are distributed over the cross section of the pipe (i.e. the flow "flow regime" or "flow pattern"). In design, it is necessary to predict the flow pattern which, usually, will depend not only on the flow behavior, but also on the superficial velocities of the phases and the distribution of the fraction occupied by each phase over the cross section of the pipe. The mean in-situ volume fraction will not normally be the same as the input volume fraction. The flow behavior is also influenced by the density and viscosity of the phases and the diameter of the pipe; studies of such parametric effects include those of Charles *et al.* (1961), Sooth and Knudsen (1972), Martinez *et al.* (1988), Arirachakaran *et al.* (1989), Urdahl *et al.* (1997), Shi and Jepson (1999). Most previous studies have concentrated on general flow patterns and their delineation through flow pattern maps. There have been only a few studies focused specifically on dispersed flows in horizontal pipelines. The present detailed understanding of this phenomena involved is very limited. In the dispersed flow region, there exist two types of flow configuration, namely: oil-in-water dispersions and water-in-oil dispersions.

A number of recent studies on oil-water dispersions have focused on horizontal pipelines and, in particular, on the evaluation of the behaviour of the droplets in the system. Extensive studies of flow patterns and the transition between them have been carried out, resulting in a better understanding of the two-phase flow structure. It is important to understand the nature of the interactions between the phases and how these influence the flow patterns and the resulting flow pattern maps, the droplet behaviour and the phase distributions. Arirachakaran *et al.* (1989), Angeli (1996) and Soleimani (2000) found that dispersed flow for oil-water systems in horizontal pipes occurs when the liquid-liquid mixture is moving at high velocity.

In horizontal flow, the flow pattern will inevitably be more complex because the gravitational force acts perpendicular to the direction of flow. Thus, there is a tendency for the dispersed phase to move vertically (i.e. normal to the tube axis) under the influence of gravity (upwards, due to buoyancy, if the dispersed phase is the lighter phase and downwards if the dispersed phase is the heavier). This tendency is affected by the action of turbulent eddies in the continuous phase which act towards making uniform the distribution of the dispersed phase due to turbulent diffusion. The actual distribution is a manifestation of the balance between gravityinduced separation and turbulence-induced mixing.

Earlier work on liquid-liquid flows in horizontal channels included studies of the phase distribution done by Angeli (1996) and Soleimani (2000). These studies demonstrated the tendency for the dispersed phase to separate to the top or the bottom of the channel depending on its density relative to the continuous phase. The higher the velocity, the more the fluids were well mixed indicating the increasing dominance of turbulence over gravity. In these earlier experiments, the measurements were made in what was expected to be a relatively fully developed flow at the end of the test section (typically 300-400 tube diameters from the inlet). However, it seemed likely that further insight could be gained regarding the turbulent mixing and gravity separation processes by studying the development of the flow along the channel and this was the underlying theme of the work reported here.

2.2 Drop break-up and Coalescence

Break-up and coalescence will determine the final drop size distribution in a dispersed flow system. The majority of the investigators have looked at these phenomena separately in an effort either to define the maximum drop diameter that can resist break-up or the collision and coalescence frequency of drops in a flow field. Some investigators have tried to combine both phenomena in models that are based on drop population balances, in order to predict the final drop size distribution in a dispersed system, but with limited experimental justification (Valentas & Amundson, 1966; Tsouris & Tavlarides, 1994). It should be noted here that most of the experimental and theoretical work on dispersed phase drop size comes from stirred vessels, and that only a few investigators have considered the breakage or coalescence of drops in a turbulent pipe flow.

2.2.1 Drop break-up

The fundamental work on drop break-up in a turbulent flow field was conducted independently by both Kolmogoroff (1949) and Hinze (1955). According to them, the force from the continuous phase will tend to deform the drop, while the interfacial tension and viscosity of the dispersed phase will tend to stabilise it. Two dimensionless groups can be formed from these forces:

- a generalised Weber number:

  $\text{We} = \pi d/\text{p}$

- a viscosity group that accounts for the dispersed phase viscosity:

$$N_{Vi} = \frac{\eta_d}{\sqrt{\rho_d \sigma d}},$$

2.2.2 Drop coalescence

For drops to coalesce in a turbulent flow field they must first collide and then remain in contact for sufficient time for the film of the continuous phase that has been trapped between the drops to drain to a critical thickness and then rupture. During this contact period, and before coalescence occurs, turbulent eddies of the continuous phase may separate the drops and prevent coalescence. The modelling of coalescence is hindered from the fact that the film drainage process can not be described easily (Valentas & Amundson, 1966; Thomas, 1981).

Shinnar (1961) considered that drop coalescence, like drop break-up, also happens in the inertial subrange of turbulence. Then two drops collide then coalescence will not happen if their kinetic energy, which will take them apart again, is larger than their adhesion energy. There will therefore be a minimum drop diameter d_{min} for which separation after collision can still happen (Shinnar, 1961) and for drops with diameters larger than d_{min}coalescence is not possible. Thomas (1981) considered a minimum contact time to be necessary for the drops to coalesce. He also reached a similar to Shinnar's conclusion, that coalescence will occur when the diameters of the colliding drops are smaller than a diameter d which depends on the critical film thickness necessary for the film rupture.

Howarth (1964) suggested that the coalescence frequency, v_{col}, of the drops should be given by the product of the collision frequency, v_{col}, and of the fraction of collisions that result in coalescence, f_{coal}-. He defined the coalescence frequency in terms of a critical velocity of approach between the two colliding drops, but gave no values for it. In general, the work on drop coalescencemay give some insight on the phenomenon, but thedifferent relationships which have been proposed cannot be readily used in practical situations, since there are noexpressions available for the necessary parameters (e.g. critical film thickness between the drops where coalescence will occur).

2.2.3 Drop size distribution

Drop size distributions during the pipe flow of two immiscible liquids have been given by a few investigators. In the work of Ward and Knudsen (1967) the dispersion was formed in a stirred tank before it entered the test section through a pump. The drops may thus have had sizes acquired in regions of high shear in the tank and the pump and not because of the turbulent action of the flow. Collins and Knudsen (1970) found that the drop size distribution comprised of two superimposed distributions, one produced by the nozzle injecting the dispersed phase, and the other produced by the turbulence of the flowing stream. Unlike the other investigators, the dispersed flow distributions given by El-Hamouz and Stewart (1996) were formed downstream of a static mixer; therefore the drop sizes recorded were mainly due to the action of coalescence. The work of Karabelas (1978) is of particular value since he observed drops formed due to turbulence within the pipeline using both a photographic and a drop encapsulating techniques. Karabelas (1978) found that both the Rosin-Rammler and the upper limit log-normal distributions described his experimental drop size distributions satisfactorily. Simmons, Azzopardi and Zaidi (1998) found that the drop size distributions formed in their 4 m long test section could be fitted better by the upper limit log-normal than the normal distribution. The Rosin-Rammler distribution is described by the following relation (Mugele & Evans, 1951):

$$1 - V_{\mathrm{cum}} = \exp\left(- \left(\frac{d}{\alpha} \right)^{\delta} \right),$$

where v_{cum}. is the cumulative volume fraction of the drops that have diameters less than d and a, δ are the parameters of the distribution. This distribution can therefore be described only by the two parameters a and δ and its simple form makes it appealing for engineering calculations. Karabelas (1978) found that for his data the parameter δ varied between 2.3 and 2.9.

An easier way to compare drop size distributions is by using characteristic mean diameters. The Sauter mean diameter, d_{32}, which is often used to characterise dispersions formed in pipelines is defined as follows:

$$d_{32} = \frac{\sum_{j=1}^{n} d_j^3}{\sum_{j=1}^{n} d_j^2},$$

where n is the number of drops in the distribution and d_j is the diameter of the drop j in a drop distribution.

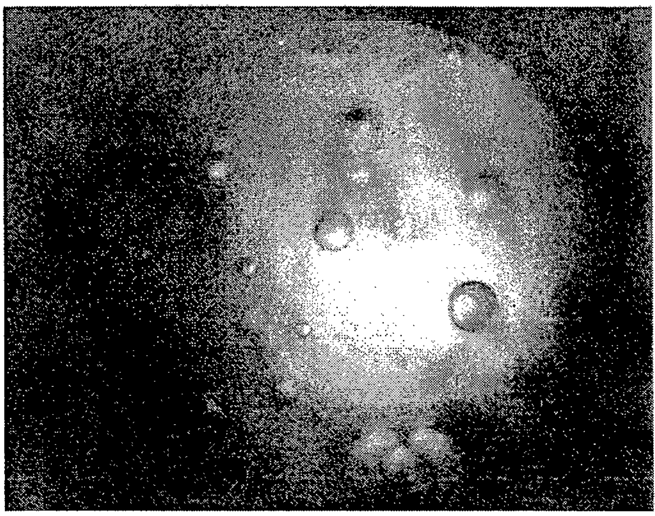

Fig. 2.1 Water drops in oil in the acrylic resin pipe. (Angeli, 2000)

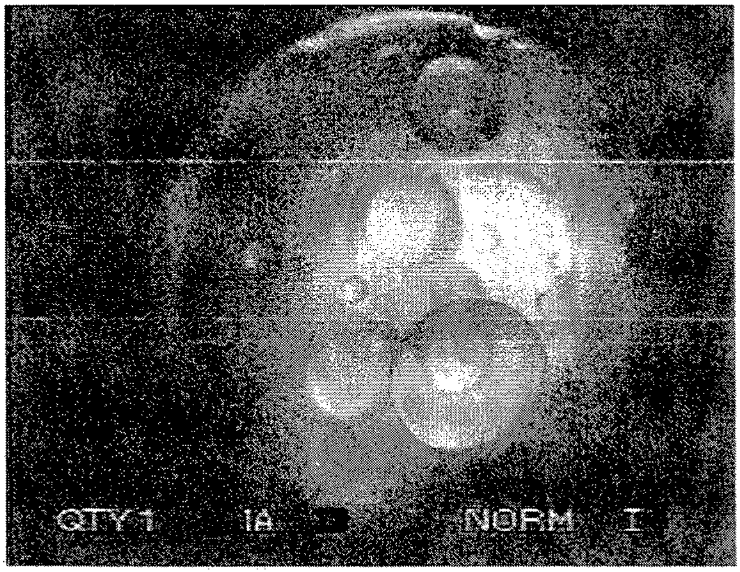

Fig. 2.2 Oli drop in water in the acrylic resin pipe. (Angeli, 2000)

2.3 Continuous and Direct experiments

Most of the experiments reported were performed in a stirred vessel and usually water and oil were used. They were often continuous experiments during which the dispersed phase was gradually added to the continuous phase. For this type of experiments it was found, that phase inversion could be postponed to a high value

(>0.8) of the dispersed phase volume fraction. Also a wide ambivalent volume fraction region existed where the mixture could be either water continuous or oil continuous (Vaessen et al., 1996; Groeneweg et al., 1998; Deshpande and Kumar, 2003; Mira et al., 2003; Tyrode et al., 2003). During direct experiments in a stirred vessel the two liquids were mixed at a certain concentration (Quinn and Sigloh, 1963; Tyrode et al., 2005) and inversion usually occurred at a value of the dispersed phase fraction close to 0.5(dependent on the properties of the liquids) and no ambivalence region was observed. It is important to point out, that in the above mentioned papers some are without an added surfactant (for instance Deshpande and Kumar, 2003; Pacek and Nienow, 1995; Liu et al., 2005, 2006) and others with an added surfactant (for instance Binks and Lumsdon, 2000; Rondo´n-Gonzale´z et al., 2006; Tyrode et al., 2005). The presence of a surfactant can have a significant influence on the inversion process; they tend to favor one type of dispersion over the other. In our experiments we have not added a surfactant. However, in the oil (that we used) there were small concentrations of substances that were acting like a surfactant, as water was favored as the continuous phase. Only a few phase-inversion experiments were carried out in a pipe.

Direct experiments were done by Liu et al. (2006) in a vertical pipe and by Pal (1993), Na}dler and Mewes (1997), Ioannou et al. (2005) and Chakrabarti et al. (2006) in a horizontal pipe. They paid particular attention to the pressure drop increase during phase inversion. Continuous experiments were performed in a horizontal pipe (Piela et al. 2006). A strong increase in the pressure drop were measured during the inversion process. Moreover pictures were made of the change in morphological oil–water structures during inversion. As for the stirred vessel experiments the comparison between the direct experiments and the continuous experiments showed, that also in the case of a pipe flow the critical concentration of the dispersed phase fraction at inversion was very different for the two types of experiments. Dependent on the experimental conditions the critical concentration can be significantly higher for continuous experiments than for direct experiments. For practical applications this result is very important, as it opens the opportunity to avoid or postpone phase inversion (causing a high pressure drop or low flow rate) by gradual injection of the dispersed phase into the continuous one. Therefore it is decided a new set of experiments is to carried out to study the critical concentration for continuous experiments as function of some parameters (such as the injection phase volume fraction). To get additional information also detailed pictures were taken during phase inversion for both continuous and direct experiments.

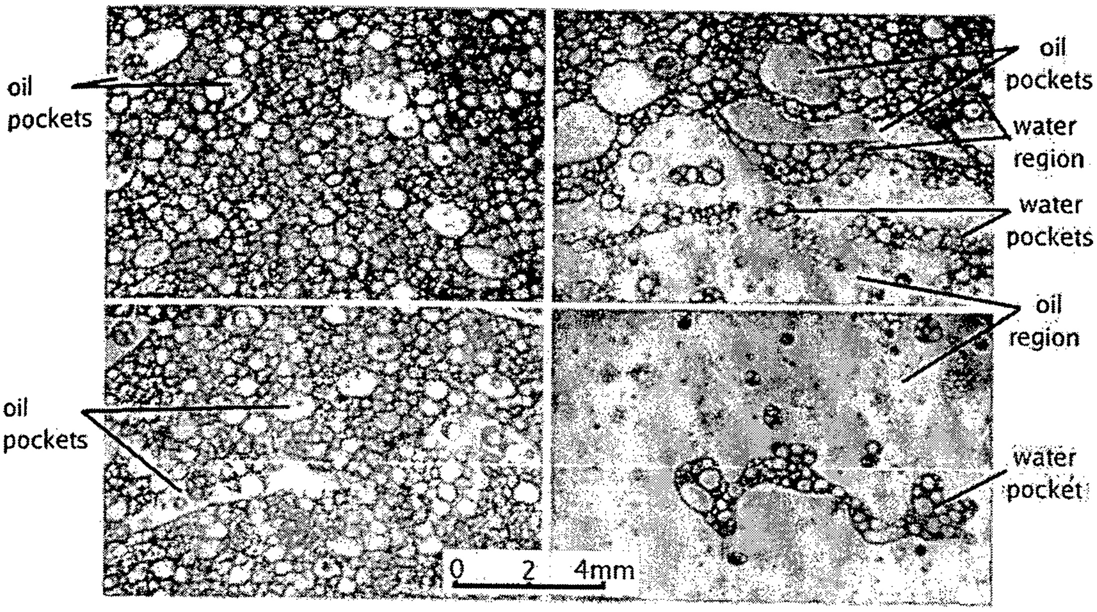

Fig 2.3 Inversion from a water-continuous flow to an oil-continuous flow during a continuous experiment. Starting from the top-left picture,the bottom-left picture is taken after 16s, the top-right after 43s and bottom-right after 82s. (Piela et al.,2008)

CHAPTER 3

MATHEMATICAL MODELLING

3.1 Overview

This chapter gives a description about modeling of the phase inversion in pipelines and the mathematical modeling of CFD approach. In modeling the phase inversion in pipeline, Gambit 2.4.6 software was used to draw the scale of the pipeline as well as to generate the computational grid. Once created, the grid is then exported to Fluent 6.3.26 software for simulation process. This chapter also described about the oil-water phase inversion modeling, the turbulence model and the CFD approach for pipeline.

3.2 Introduction

In this study, the field equations have been solved using FLUENT (Version 6.2). The 'quadrilateral' cells of non-uniform grid spacing were generated using the commercial grid tool GAMBIT. The 3-D and unsteady solver was used to solve the incompressible flow on the collocated grid arrangement. The second order upwind scheme was used to discretize the convective terms in the momentum and energy equations. The semi-implicit method for the pressure linked equations (SIMPLE) scheme was used for solving the pressure-velocity decoupling. The constant density and Eulerian–Eulerian model were used to compute turbulent dispersed oil–water two-phase flow in a horizontal pipe, where the standard $k-\varepsilon$ turbulent model was used. Interphase forces such as drag, lift and turbulent dispersion forces were included in the present study. FLUENT solves the system of algebraic equations using the Gauss-Siedel (G-S) point-by-point iterative method in conjunction with the algebraic multi-grid (AMG) method solver. The use of AMG scheme can greatly reduce the number of iterations (and thus, CPU time) required to obtain a converged solution, particularly when the model contains a large number of control volumes. Relative convergence criteria of 10−4were prescribed in this work and solution converged when all the residuals reached specified convergence criterion. Grid independence study was carried out with four different grids and a grid consisting approximately1,60,000 cells is used in this work as shown in Fig. 2. The time dependent simulations are performed with time step size of 0.1 s to achieve numerical stability.