CFD SIMULATION OF OIL-WATER FLOW IN A HORIZONTAL PIPE

MUHAMMAD SYAHMI AFIF BIN MOKHTAR YAZID

A project report submitted in partial fulfilment of the
requirements for the award of the degree of
Master of Petroleum Engineering

School of Chemical and Energy Engineering
Faculty of Engineering
Universiti Teknologi Malaysia

JANUARY 2019

DEDICATION

This thesis is dedicated to my beloved family and friends for their love, support and encouragement.

ACKNOWLEDGEMENT

In preparing this thesis, I was in contact with many people, researchers, academicians, and practitioners. They have contributed towards my understanding and thoughts. Thanks to Almighty God for giving me the strength, wisdom and understand to successfully complete my simulation work. In particular, I wish to express my sincere appreciation to my thesis supervisor, Associate Professor Issham Ismail, for his encouragement, guidance, critics, brilliant ideas, suggestions and friendship.

My sincere appreciation goes out to my parent and siblings for their support and encouragement. Also I am thankful to my elder brother for his useful advice and love.

I would like to extend my gratitude to all the academic staff and non-academic staff for their support towards the success of this simulation work. I am also very thankful to all my friends for their advices and motivation. Without their continued support and interest, this thesis would not have been the same as presented here. Their views and tips are useful indeed. Unfortunately, it is not possible to list all of them in this limited space but I would like to let them know that I really appreciate all their kindness.

ABSTRACT

Water produced along with crude oil during production and transported together in a pipeline is a common occurrence in a petroleum production system. Understanding the behaviour of crude oil-water flow in a pipeline is crucial to many engineering applications, such as design and operation of flowlines and wells, separation systems, logs interpretations and determination of the amount of free water in contact with the wall of the pipes that can render erosion or corrosion problems. Presently, there are only two published papers on two phase flow of Malaysia waxy crude oil-water, namely Ismail *et al.* (2015b) and Piroozian *et al.* (2017). Their focuses were on flow pattern, pressure drop, and water holdup of the two phase flow system. All the parameters, working conditions, and simulation for this research work were basically based on the experimental results published by Ismail *et al.* (2015b) which involved the flow of waxy crude oil-water in a closed-loop system of 5.08 cm ID stainless steel horizontal pipeline at ambient condition. The water cut was varied from 10% - 90% with mixture velocities ranging from 0.1 – 0.9 m/s. Five flow patterns have been identified by them; stratified waxy flow, stratified wavy semi dispersed flow at interface and oil film, dispersion of water in oil and oil continuous with emulsion, dispersion of oil in water with water continuous, and the newly found semi dispersed flow semi emulsion at interface and thin oil film. The objective of this study is to simulate and validate the flow pattern, pressure drop and liquid holdup with the experimental results using ANSYS Fluent and CFX-Pre. The volume of fluid (VOF) multiphase flow modeling method was used in Fluent and free surface model was used in CFX-Pre in conjunction with the standard k-ε and k-ω schemes to simulate the oil-water flow. The simulation results were found to be in good agreement with the published data. The simulation results can be used as a platform to understand better other more complex cases of waxy crude and water concurrent flow in pipelines.

ABSTRAK

Pengeluaran air bersama dengan minyak mentah ketika telaga berpengeluaran dan penghantaran melalui talian paip ialah suatu kebiasaan bagi sistem pengeluaran petroleum. Kefahaman tentang perilaku aliran minyak mentah-air di dalam talian paip adalah penting untuk pelbagai pengaplikasian kejuruteraan, misalnya mereka bentuk dan pengoperasian talian aliran dan telaga, sistem pemisahan, pentafsiran log dan penentuan jumlah air bebas yang bersentuh dengan dinding paip, yang boleh menimbulkan masalah karatan atau hakisan. Sehingga kini, hanya dua kumpulan penyelidik yang telah menerbitkan artikel tentang aliran dua fasa yang mengguna pakai minyak mentah berlilin Malaysia, iaitu Ismail *et al.* (2015b) dan Piroozian *et al.* (2017). Kajian ini mereka tertumpu terhadap corak aliran, kejatuhan tekanan dan air tertahan bagi sistem aliran dua fasa. Semua parameter, keadaan uji kaji dan penyelakuan untuk kajian ini adalah berdasarkan hasil kajian makmal daripada Ismail *et al.* (2015b). Mereka mengalirkan minyak mentah berlilin di dalam sistem gelung tertutup melalui paip keluli mendatar yang berdiameter 5.08 cm pada keadaan ambien. Dalam kajian ini, potong air telah diubah daripada 10 hingga ke 90% dengan halaju campuran daripada 0.1 m/s hingga ke 0.9 m/s. Lima corak aliran telah dikenal pasti, iaitu aliran berlapis berombak, aliran berlapis berombak dengan terselerak separuh di antara muka dan lapisan minyak, aliran air terselerak di dalam minyak dengan minyak berterusan bersama-sama emulsi, aliran minyak terselerak di dalam air dengan air berterusan, dan corak aliran yang baharu ditemui iaitu aliran terselerak separuh dengan emulsi separuh di antara muka dan lapisan minyak yang nipis. Tujuan utama kajian ini adalah untuk menyelaku dan mengesahkan corak aliran, kejatuhan tekanan dan air tertahan dengan keputusan eksperimen menggunakan ANSYS Fluent dan CFX-Pre. Kaedah pemodelan isi padu cecair aliran pelbagai fasa dalam Fluent dan model permukaan bebas dalam CFX-Pre telah digunakan bersama dengan skim k-ε dan k-ω untuk menyelaku aliran minyak mentah-air. Hasil penyelakuan didapati menyerupai data artikal terbabit. Hasil kajian ini boleh digunapakai sebagai asas untuk memahami dengan lebih baik tentang kes yang lebih rumit, misalnya aliran gas, minyak dan air secara bersama di dalam talian paip.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CAD	-	Computer-Aided Design
CFD	-	Computational Fluid Dynamics
ENIAC	-	Electronic Numerical Integrator and Computer
FEM	-	Finite Element Method
FS	-	Free Surface
SIMPLE	-	Semi-Implicit Method for Pressure Linked Equations
PRESTO	-	Pressure Staggering Option
NSE	-	Navier-Stokes Equation
VOF	-	Volume of Fluid
S	-	Stratified
MO	-	Mixed Oil
MW	-	Mixed Water
AO	-	Annular Oil
AW	-	Annular Water
IO	-	Intermittent Oil
IW	-	Intermittent Water
DO	-	Dispersed Oil
DW	-	Dispersed Water
SS	-	Stratified Smooth
SW	-	Stratified Waxy
SM	-	Stratified Mixed
DC	-	Dual Continuous
DW/O&O	-	Dispersion of Water in Oil and Oil Layer
DO/W&W	-	Dispersion of Oil in Water and Water Layer
FD	-	Fully Dispersed
STW	-	Stratified Wavy Flow
STSD&O	-	Stratified Wavy Semi Dispersed Flow at Interface & Oil Film
SDSE&TO	-	Semi Dispersed Flow Semi Emulsion at Interface & Thin Oil
DWE	-	Dispersion of Water in Oil and Emulsion
DO	-	Dispersion of Oil in Water with Water Continuous

LIST OF SYMBOLS

V_m	-	Mixture velocity
ρ_m	-	Mixture density
Re	-	Reynolds number
ID	-	Internal diameter
OD	-	Outside diameter
k-ω	-	Kinetic energy and kinetic energy to turbulence energy
k-ε	-	Kinetic energy and kinetic energy to thermal energy
ΔP	-	Pressure drop
H_w	-	Water hold up
A_o	-	Area of oil
A_w	-	Area of water
S_o	-	Superficial oil
S_w	-	Superficial water
A_I	-	Superficial Interface
α_o	-	Volume fraction of oil
α_w	-	Volume fraction of water
F	-	Surface tension
κ	-	Curvature
n	-	Surface normal
G_k	-	Turbulence kinetic energy due to mean velocity gradient
G_b	-	Turbulence kinetic energy due to buoyancy
Y_M	-	Contribution of the fluctuating dilation in compressible turbulence to the overall dissipation rate
$C_{1\varepsilon}, C_{2\varepsilon}, C_{3\varepsilon}$	-	Constant
σ_k	-	Turbulent Pandtl number for k
σ_ω	-	Turbulent Pandtl number for ε
S_k and S_ω	-	User-defined sources terms

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Introduction

Crude oil and natural gas consist of two main elements which are hydrogen and carbon and they occur naturally in subsurface zone. Crude oil is a complex mixture of organic compound and hydrocarbon molecules and its name depends on the number and isomer of carbon and hydrogen atoms. The crude oil can be refined by undergoing fractional distillation in a refinery. This process will break down the liquid into various products based on its liquid compositions and temperature (Pei *et al.*, 2015).

Oil and gas has been the primary resources for energy worldwide to generate electricity, as fuel to cars, to heat buildings, etc. But the oil and gas industry is still facing major challenges in terms of processing facilities, production and transportation of the oil and gas to ensure the economical hydrocarbon flow in pipelines from deep water fields to the host platform. High water cut oil wells need to be diagnosed and treated in order to rejuvenate their oil production which can contribute positively to safe and economical operation of production facilities (Ismail *et al.*, 2015a). Due to decreasing in conventional light crude oil reserves, and existence of lots of mature oilfields around the globe, the research trend in oil and gas industry has shift towards the understanding of liquid-liquid behaviour due to existence of oil and water concurrent flow in pipeline as oil and water separation units cannot separate them with 100% efficiency (Liu *et al.*, 2011). An oil field become mature when production has reached its peak and starts to decline. At this states, reservoir pressure declines due to the depletion of the oil zone and insufficient of water influx from aquifer.

Generally mature oil wells are found to have been producing with high water cut. Nevertheless, onshore mature oil wells with high water cut can be economical and reasonable to operate based on the current oil price (Parshall, 2012). On top of it, about 2/3 of the world oil productions mainly comes from mature oil fields and only 1/3 comes from new field. Besides, the conventional light crude oil field is estimated to have half of the global oil reserve. As most of the oil fields worldwide contain basically light crude oil reservoirs, their depletion in future will give rise to an extensive study for the heavy crude oil for future energy sources (Luppens, 1995).

In recent decades, there are vast amount of research studies relating to the gas-liquid behaviour such as McNulty *et al.* (1992) and Das *et al.* (1999) but only few research works have been made for liquid-liquid behaviour such as Sridhar *et al.* (2011) due to its uniqueness, complexity, complicated rheological behaviour and huge difference in pressure gradient for different flow patterns (Arirachakaran *et al.*, 1989). Basically, they are mainly focusing on flow pattern, pressure drop and liquid hold up (Yu *et al.*, 2010). The understanding of these behaviours is important for many engineering applications including production logs interpretations, flow lines and wells operations and monitoring and designing of separation processes (Atmaca *et al.*, 2009).

In multiphase flow, there are four main types of two phase flow which are gas-liquid, gas-solid, liquid-liquid, and liquid-solid. According to Soleimani *et al.* (2000), they stated that gas-liquid is the combination of characteristics of a deformable interface and the compressibility of one of the phases. They also mentioned that there are five flow regimes for vertical flow which are bubble, slug or plug, churn, annular and wispy annular flow, whereby there are six flow patterns for horizontal flow such as dispersed bubble, stratified, stratified-wavy, plug, slug, and annular-dispersed flow. The detailed explanation for each flow will be discussed in Chapter 2. There are immense research works on gas-liquid two phase flow which focus on the flow of fluids inside the boiler, nuclear reactors, refineries, artificial lift systems, crude oil transportation, petrochemicals, nuclear reactor, etc. as described by Ismail *et al.* (2015a), Brauner (2002) and Flores *et al.* (1999).

Numerous two phase flow studies done in the past which covered flow patterns by Yu *et al.* (2010) and McNulty and Sutjipto (1992), physical mechanism by Das *et al.* (1999), pressure gradient by Yuan and Zhou (2009), pressure drop and liquid hold up by Yu *et al.* (2010). But due to majority of the developed correlations and mechanical models are based on certain specific area or conditions, it is hard to determine on which correlation is the most accurate and can predict the precise outcomes.

Liquid-liquid flow can be defined as the flow of two immiscible liquids in a given pipe that flow simultaneously which is basically oil and water. Arirachajaran (1989) stated that the nature of the liquid-liquid flow is highly complex and complicated with full stratification occurring at low velocities and full dispersion at high velocities. Therefore prediction for this flow is a big challenge. This two phase flow phenomenon can be highly found on food and petrochemical industries (Das *et al.*, 2000).

As mentioned above, the primary study on liquid-liquid flow is more to determine the flow patterns, pressure drop and liquid hold up, however, any changes in hydrodynamics behaviour caused by differences in physical and chemical properties and extensive range of operating conditions lead to large complication in generalizing and standardizing the liquid-liquid flow system. Moreover, the correlation and mechanical model developed still remain unsolved especially for high viscosity oil (Ismail *et al.*, 2015a). There are lots of flow pattern classifications by previous researchers on liquid-liquid flow, such as Kumara *et al.* (2009) who stated that there are two categories for flow patterns which are segregated and dispersed flow where each category has different flow patterns. Whereby, Morgan *et al.* (2013) described four flow pattern categories which are stratified, mixed, dispersed and two-layer, dispersed over/under continuous flow. Basically, in horizontal pipe, the common flow patterns can be found are stratified, mixed oil and water, annular oil and water, intermittent oil and water and dispersed oil and water (Arirachakaran *et al.*, 1989). The detailed explanation for each flow is given in Chapter 2.

Furthermore, the main difference between gas-liquid and liquid-liquid flow is gas-liquid flow involves in high viscosity and high density differences, whereas liquid-liquid system involves in low density difference as stated by Brauner (2002) and Jana *et al.* (2006). Oddie *et al.* (2003) mentioned that another major difference between these flows is slug flow which frequently occurs in gas-liquid flow but rarely observed in liquid-liquid flow. Whereby, Shi *et al.* (2003) explained that the significant difference between gas-liquid and liquid-liquid flow is the physical properties that govern them which refer to density and viscosity of both phases. An emulsion is formed when two immiscible liquids especially for oil-water that flow simultaneously in a pipeline. An emulsion can completely change the physical properties of the liquids. The emulsion is unstable and there is a natural tendency for them to separate back to their original form after some period of time (Sridhar *et al.*, 2011).

In most engineering applications, the commercial computational fluid dynamics (CFD) has become popular with advancement of computer technologies as it is a useful tool that can give insight and foresight of the flow behaviours. To perform the CFD simulation correctly, one needs to have a really good understanding on the CFD model (Gidaspow *et al.,* 2013). Moreover, CFD simulation can provide opportunity to simulate a complex and complicated real flow especially when experimental studies become extremely demanding. Based on a recent study, the CFD simulation has the ability to simulate two phase flow such as solid-liquid and gas-liquid in an annulus in reference to axial fluid, solid velocities, pressure drop profile, rate of penetration, inclination of pipe, and fluid rheological behaviour (Epelle and Gerogiorgis, 2017).

There are several CFD solver packages have been developed for industrial usages and research studies. The popular CFDs are ANSYS Fluent and NEPTUNE CFD. Shi *et al.* (2017) studied the flow patterns, pressure drop and water hold up for oil-water phase in a horizontal pipe with medium viscosity using ANSYS Fluent package, whereas Mimouni *et al.* (2017) studied the flow pattern map for multiphase flow using NEPTUNE CFD package.

1.2 Problem Statement

In recent oil-water two phase flow studies, several correlations, mechanical and empirical model has been developed by many researcher such as Arirachakaran *et al.* (1989) and Bannward (1998) but it is a great challenge to determine which model would give the most accurate and reliable result because each model is based on certain specific area or conditions including assumptions especially for high viscosity crude oil. Moreover, as stated by Trallero *et al.* (1997) and Jana *et al.* (2012), in order to develop these models, a careful analysis for the flow behaviour is required such as physical properties effect on the fluids, stability of the flow, materials of the pipe used, transition diameter and criteria, drop size distribution and momentum transfer capacity effect. Thus, a comprehensive research work needs to be done in order to get precise concept and mechanism of the oil-water flow. Moreover, there are vast amount of research works that have been done for low and medium viscosity crude oil such as by Poesio *et al.* (2008), Grassi *et al.* (2008), and Sunder Raj *et al.* (2005) but for high viscosity crude oil the research is limited. Some of the researchers tried to use their previous low viscosity oil model to develop high viscosity model like Grassi *et al.* (2008) and Jana *et al.* (2006), but as the liquid-liquid flow behaviour is highly characterized by the viscosity ratio, therefore using low viscosity oil model to represent high viscosity oil model is not a straight forward step because it involves some parameters that could not be predicted.

Furthermore, there are some factors that highly contributed to the formation of flow pattern of the liquid-liquid flow such as rheological behaviour, pressure gradient difference, position of the pipe (e.g., horizontal, vertical, and inclination), temperature, mineral oil versus crude oil and ease to form emulsion (Arirachakaran, 1989). The flow of oil-water most likely tends to form an emulsion and it may exhibit a Newtonian or non-Newtonian rheological behaviour that highly dependent on oil-water system (e.g., flow pattern, pressure drop, and liquid hold up). The pressure gradient increases with oil and water superficial velocity and it is mainly influenced by the viscosity of the oil as well as the effect of flow patterns transition. The gravitational effect for the position of the pipe such as vertical and inclination could contribute to pressure drop difference.

Moreover, if the temperature of a system falls below the wax appearance temperature (WAT), the precipitation of paraffin crude oil will occur and this can clog the pipe. There is a significant difference between mineral and crude oil in oil-water system experiment due to their composition differences. As the crude oil comprises natural surfactant molecules such as resins, asphaltenes, organic acids, and waxes they might be easy to form emulsion rather than mineral oil (Ismail *et al.*, 2015a). As the emulsion formed is quite stable, thus it contributes to more complex and complicated oil-liquid flow system. Crude oil is the oil that comes from the earth crust whereas mineral oil is resulted from crude oil that has undergone fractional distillation.

Sometimes, when the experimental data has become extremely complicated, CFD simulation is the best choice as it uses numerical analysis and data structure to solve and analyze complex problems that involve fluids flow. Other major advantages of CFD are it can foresee performance of system, determine the change in design layout and provide detailed information for heating parameter and flow pattern within an occupied zone. In CFD, there are few methods which can be used to simulate the flow depending on the phase of the fluids and normally the most common way to solve it is by using discretization method. Discretization method is a method of a process of transferring continuous model, variables and equations into discrete. This discretization method is obtained from derivation of finite element model which is a numerical method that uses approximation to solve the equations. When running the CFD simulation, the basic equation use is Navier-Stokes equations that encompassed another three equations which are continuity, momentum and energy. For simulation of two phase turbulent flow in CFD, another two equations should be added together with Navier-Stokes such as volume of fluid (VOF) method for ANSYS Fluent or free surface model for ANSYS CFX-Pre and turbulence model. The VOF is considered under the Eulerian method where it happens in control volume (i.e., each molecule is expressed in a function of space and time and velocity behaviour in the particular time). Those models were studied in order to determine the best-fit one for the selected published data. The experimental data of flow pattern, pressure drop and water hold up by Ismail *et al.* (2015b) was used to compare the experimental and simulation results using ANSYS Fluent and CFX-Pre.

1.3 Research Objectives

The objectives of this project are as follow:

(1) To determine which method and model of CFD can be used to simulate the flow pattern in ANSYS Fluent.

(2) To model and simulate the flow pattern, pressure drop and water hold up of the West Malaysian mild waxy crude oil for a given set of flow rates in a closed flow loop system at ambient condition in horizontal pipe.

(3) To compare the simulation result with experimental result produced by Ismail *et al.* (2015b).

1.4 Research Scope

The research work was initiated by defining the facilities. In the experimental studies, the length of the pipe used was 49.27 m with 5.08 cm inside diameter of pipe. For the simulation work, only 2 m pipe length was taken for this study which was in the visualization box section and 5.08 cm inside diameter. AutoDesk Inventor was used to model the geometry then imported to ANSYS Fluent for mesh generation. Three different types of mesh were studied and the best mesh was tetrahedron mesh for this simulation work.

The next step was determining which method and model could be used in oil-liquid phase flow system. It is widely known that some of the models can only work on liquid-liquid phase but fail for gas-solid and gas-liquid system. Even though most of the methods and models can be used in modelling and simulation of the oil-liquid phase, but to determine the best-fit will be a real time consuming and challenging as we need to really understand their detailed working principles and mechanisms.

Next, after choosing the best method, the CFD model and simulation setup was created. The first step is modelling the method chosen by considering an isothermal system with no mass transfer and no phase change by stating the conservation equations such as their mass and momentum equations. Then the interface between the water phase (primary) and oil phase (secondary) needs to be tracked by solving the conservation equation for the volume fraction of the secondary phase (oil). After that, boundary and initial conditions of the model such that velocity for inlet boundary is set up at the inlets. Setup for the model needs to done exactly which involves the experimental parameters such as velocity, density, viscosity, oil and water fraction, surface tension etc. to determine flow pattern, pressure drop and liquid hold up. Lastly, the flow pattern map generated by ANSYS Fluent for the waxy crude oil-water system is compared with ANSYS CFX-Pre and experimental data for validation purposes.

1.5 Significance of Study

The main reason for this study is to enhance the knowledge and understanding on the flow pattern, pressure drop and liquid hold up for the oil-water two phases flow by using ANSYS Fluent and CFX-Pre solver package. To achieve it, equations involve in fluid mechanics need to be fully understood such as Navier-Stokes equation in order to determine the best-fit model that can be used for the simulation work. As most researchers and industry practitioners are likely to use the popular CFD solver such as ANSYS Fluent and NEPTUNE CFD, none of the research work has simulated waxy oil-water two phase flows in pipeline using any CFD simulation. Thus, lack of information for this condition is the greatest obstacle and therefore a deeper study needs to be done in order to achieve an accurate and reliable result. Furthermore, the result of this research work can provide an advanced knowledge of CFD simulation especially for ANSYS Fluent and CFX-Pre and it can be used as a reference for further study on two phase flow system or for other engineering applications such as heating, ventilation, production logs interpretation, and well operations or interventions.

1.6 Chapter Summary

This chapter briefly describes an oil-water two phase flow systems, its definition and importance to the petroleum industry. It is complemented with the problems and difficulties encountered in a two phase flow system including the operational and designing part of offshore facilities and gaps between previous studies. The need to study the Malaysian waxy crude oils behaviour in a two phase flow system was also highlighted. To addressing these issues, the objectives and scopes of the research work were outlined. This chapter is complemented with the significance of this research work to the Malaysia's petroleum industry. Moreover, the application of commercial CFD and its usage in industry is also describes in this chapter. One of the important of CFD simulation is when the experimental data has become extremely complicated, CFD simulation is the best choice as it uses numerical analysis and data structure to solve and analyse complex problems that involve fluids flow. Furthermore, a briefly explanation about the simulation of two phase flow in a horizontal pipe also being discussed and further details on steps to do the simulation is provided in Chapter 3.

REFERENCES

Alipchenkov, V. M., Nigmatulin, R. I., Soloviev, S. L., Stonik, O. G., Zaichik, L. I. and Zeigarnik, Y. A. 2004. A three-fluid model of two-phase dispersed-annular flow. *Int. J. Heat and Mass Transfer*. 47(24), 5323-5338.

Arirachakaran, S., Oglesby, K. D., Malinowsky, M. S., Shoham, O. and Brill, J. P. 1989. An analysis of oil/water flow phenomena in horizontal pipes. SPE-18836. *Presented at the SPE Production Operation Symposium*, Oklahoma, U.S.A.

Asif, M. 2010. Stratification pattern in inverse binary-solid liquid-fluidized beds. *Int. J. Mineral Processing*. 97(1-4), 79-84.

Atmaca, S., Sarica, C., Zhang, H. Q. and Al-Sarkhi, A. S. 2009. Characterization of oil/water flows in inclined pipes. SPE-115485-PA-P. *Presented at the SPE Annual Technical Conference and Exhibition,* Denver, U.S.A.

Ayello, F., Li, C., Tang, X., Cai, J. and Nesic, S. 2008. Determination of phase wetting on oil-water pipe flows. Paper no 08566 presented in *NACE International Corrosion Conference & Exposition*, Houston, U.S.A.

Bannwart, A. C. 1998. Wave speed and volumetric fraction in core annular flow. *International Journal of Multiphase Flow*. 24, 961-974.

Bestion, D. 2014. The difficult challenge of a two phase CFD modeling for all flow regimes. *Nuclear Engineering and Design*. 279, 116-125.

Bhutta, M. M., Hayat, N., Bashir, M. H., Khan, A.R., Ahmad, K. N., and Khan, S. 2012. CFD applications in various heat exchangers design: A review. *Applied Thermal Engineering*. 32, 1-12.

Bombardelli, F. A., Cantero, M. I., Buscaglia, G. C., & García, M. H. (2004). Comparative study of convergence of CFD commercial codes when simulating dense underflows. *Mecánica computacional*, *23*, 1187-1199.

Bordalo, S. N. and Oliveira, R. C. 2007. Experimental study of oil/water flow with paraffin precipitation in subsea pipeline. SPE 110810. Presented at the *SPE Annual Technical Conference and Exhibition*, Anaheim, U.S.A.

Brackbill, J., Kothe, D. B., Zemach, C. 1992. A continuum method for modelling surface tension. *Journal of Computer Physics.* 100 (2), 335-354.

Brauner, N. 1991. Two phase liquid-liquid annular flow. *Int. J. Multiphase Flow.* 17(1), 59-76.

Brauner, N. 2002. Modeling and control of two phase phenomena: liquid–liquid two phase flow system. *CISM Center.* School of Engineering, Tel-Aviv University, Udine, Italy.

Brennen. C. E. 2005. *Fundamentals of multiphase flows.* (First edition). Oxford. Oxford University Press, Inc.

Charles, M. E., Govier, G. W. and Hodgson, G. W. 1961. The horizontal pipeline flow of equal density oil-water mixture. *Can. J. Chem. Eng.* 39, 27-29.

Collier, J. G. and Thome, J. R. 1994. Convective Boiling and Condensation, third ed., *Oxford University Press*, Oxford.

Crowe, C. T. 2000. On models for turbulence modulation in fluid-particle flows. *Int. J. Multiphase Flow.* 26(5), 719-727.

Das, G., Das, P. K., Purohit, N. K. and Mitra, A. K. 1999. Flow pattern transition during gas liquid upflow through vertical concentric annuli. I. Experimental investigation, ASME Trans. *J. Fluid Eng.* 122, 895-991.

Das, G., Das, P. K., Purohit, N. K. and Mitra, A. K. 2000. Phase distribution of gas–liquid mixtures in concentric annuli-inception and termination of asymmetric. *Int. J. Multiphase Flow.* 26, 857–876.

Dehkordi, P. B., Colombo, L. P. M., Guilizzoni, M. and Sotgia, G. 2017. CFD simulation with experimental validation of oil-water core-annular flows through Venturi and Nozzle flow meters. *Journal of Petroleum Science and Engineering.* 149, 540-552.

Desamala, A. B. Vijayan, V. Dasari, A. Dasmahapatra, A. K. and Mandal, T. K. 2016. Prediction of oil-water flow patterns, radial distribution of volume fraction pressure and velocity during separated flows in horizontal pipe. *Journal of Hydrodynamics.* 28 (4), 658-668.

Desjardins, O. 2014. Two-phase flows – overview. *Proceeding of The Summer Program.* Pp. 57.

Dong, K. J., Guo, B. Y., Chu, K. W., Yu, A. B. and Brake, I. 2008. Simulation of liquid-solid flow in a coal distributor. *Minerals Engineering.* 21(11), 789-796.

Elseth, G. 2001. *An experimental study of oil-water flow in horizontal pipes.* Ph.D Thesis. Norwegian University of Science and Technology.

Epelle, E. I. and Gerogiorgis, D. I. 2017. A multiparametric CFD analysis of multiphase annular flows for oil and gas drilling applications. *Computers and Chemical Engineering*. 106, 645–661.

Fabre, J. and Line, A. 1992. Modeling of two-phase slug flow. *Annu. Rev. Fluid Mech*. 24, 21-46.

Farge, M. and Schneider, K. 2001. Coherent Vortex Simulation (CVS), A Semi-Deterministic Turbulence Model Using Wavelets. *Flow, Turbulence and Combustion*. 66 (4), 393–426.

Ferrari, A., Martinez, J. J., Borgne, T. L., Meheust, Y. and Lunati Ivan. 2015. Challenges in modelling unstable two-phase flow experiments in porous micromodels. *Water Resources Research*. 51(3), 1381-1400.

Flores, J. G., Chen, X. T., Sarica, C. and Brill, J. P. 1999. Characterization of oil/water flow pattern in vertical and deviated wells. SPE-56108. *J. SPE Prod. Facil*. 14 (2), 102–109.

Geuzaine, C. and Remacle, J. F. 2009. *A three-dimensional finite element mesh generator with built-in pre- and post-processing facilities. International Journal for Numerical Methods in Engineering*. 79 (11), 1309–1331.

Ghiaasiaan, S. M. Two phase flow, boiling and condenstaion: In conventional and miniature systems. *Cambridge University Press*. Cambridge, pp. 362.

Gidaspow, D., Li, F. and Huang, J. 2013. A CFD simulator for multiphase flow in reservoirs and pipes. *Powder Technology*. 242, 2–12.

Goldstein, D. and Vasilyev, O. 1995. Stochastic coherent adaptive large eddy simulation method. *Physics of Fluids A*. 24 (7), 2497.

Grassi, B., Strazza, D. and Poesio, P. 2008. Experimental validation of theoretical models in two-phase high-viscosity ratio liquid–liquid flows in horizontal and slightly inclined pipes, *Int. J. Multiphase Flow*. 34, 950-965.

Gupta, R., Fletcher, D. F. and Haynes, B. S. 2009. On the CFD modelling of taylor flow in microchannels. *Chemical Engineering Science*. 64 (12), 2941-2950.

Guzhov, A. I. and Medredev, O. P. 1971. Pressure losses in flow of two mutually immiscible liquids. *Int. Chem. Eng*. 11, 104-106.

Guzhov, A. I., Grishin, A. D., Medredev, V. F. and Medreva, O. P. 1973. Emulsion formation during the flow of two immiscible liquids. *Nefts. Choz*. 8, 58-61Hetsroni, G., Mosyak, A., Pogrebnyak, E. and Yarin, L. P. 2005. Fluid flow in micro-channels. *Int. J. Heat and Mass Transfer*. 48(10), 1982-1998.

Hess, J. L. and Smith, A. M. O. 1967. Calculation of potential flow about arbitrary bodies. *Progress in Aerospace Sciences*. 8, 1-138.

Hewitt, F. 2012. Churn and wispy annular flow regimes in vertical gas − Liquid Flows. *Energy & Fuels*. 26 (7), 4067-4077.

Huebner, K. H., Thornton, E. A. and Byron, T. D. 1995. *The Finite Element Method for Engineers* (Third ed.). Wiley Interscience.

Hughmark, G. A. 1971. Drop breakup in turbulent pipe flow. *AlChe. J.* 17, 1000.

Ismail, A. S. I., Ismail, I. & Zoveidavianpoor, M., Mohsin, R., Piroozian, A., Misnan, M. S. and Sariman, M. Z. 2015a. Review of oil – water through pipes. *Flow Measurement and Instrumentation*. 45, 357–374.

Ismail, A. S. I., Ismail, I. & Zoveidavianpoor, M., Mohsin, R., Piroozian, A., Misnan, M. S. and Sariman, M. Z.2015b. Experimental investigation of oil – water two-phase flow in horizontal pipes : Pressure losses , liquid holdup and flow patterns. *Journal of Petroleum Science and Engineering*. 127, 409–420.

Jameson, A., Schmidt, W. and Turkel, E.1981. Numerical solution of the Euler equation by Finite Volume Methods using Runge-Kutta time stepping schemes. *AIAA Journal*. 1-19.

Jana, A. K., Das, G. and Das, P. K. 2006. A novel technique to identify flow patterns during liquid–liquid two-phase up flow through a vertical pipe, *J. Ind. Eng. Chem. Res.* 45, 2381-2393.

Jayanti, S. and Brauner, N. 1992. Churn flow. Proc. Third Int. *Workshop on Two-Phase Flow Fundamentals,* London, UK, June 15-19.

Kataoka, I. and Serizawa, I. 1990. Interfacial area concentration in bubbly flow. *Nuclear Engineering and Design*. 120, 163-180.

Kawaguti, M. 1953. Numerical solution of the NS equations for the flow around a circular cylinder at Reynolds number 40. *Journal of Phy. Soc. Japan*. 8, 747-757.

Kazi, S. N. 2012. An overview of heat transfer phenomena. (First edition). Rijeka, Croatia. InTech.

Kloss, C., Goniva, C., Hager, A., Amberger, S., & Pirker, S. (2012). Models, algorithms and validation for opensource DEM and CFD–DEM. *Progress in Computational Fluid Dynamics, an International Journal*, *12*(2-3), 140-152.

Kumara, W. A. S., Halvorsen, B. M. and Melaaen, M. C. 2009. Pressure drop, flow pattern and local water volume fraction measurements of oil-water flow in pipes. *Mesurement Science and Technology*. 20, 11.

Kutta, R. 1981. Time-stepping schemes. Presented at the *AIAA 14th Fluid and Plasma Dynamics Conference*, Palo Alto, California.

Launder, B. E. and Spalding, D. B. 1972. Lectures in Mathematical Models of Turbulence. Presented at *Academic Press*, London, England.

Lim, H., Yu, Y., Glimm, J., Li, X. L., & Sharp, D. H. (2008). Chaos, transport and mesh convergence for fluid mixing. *Acta Mathematicae Applicatae Sinica, English Series*, *24*(3), 355-368.

Liu, Y., Cui, J. and Li, W. Z. 2011. A two-phase, two-component model for vertical upward gas–liquid annular flow. *International Journal of Heat and Fluid Flow*, 32 (4), 796–804.

Loseille, A., Dervieux, A., Frey, P., & Alauzet, F. 2007. Achievement of global second order mesh convergence for discontinuous flows with adapted unstructured meshes. In *18th AIAA Computational Fluid Dynamics Conference* (p. 4186)

Lun, I., Calay, R. K. and Holdo, A. E. 1996. Modelling two-phase flows using CFD, *Applied Energy*, 53 (3), 299-314.

Lundgren, T.S. 1969. Model equation for nonhomogeneous turbulence. *Physics of Fluids A*. 12 (3), 485–497.

Luppens, J.C. 1995. Practical automation for mature producing areas. SPE-26591-PA. *SPE Comp App*. 7 (2), 44-48.

Martinez, A. E., Intevep, S. A., Arirachakran, S., Shoham, S. and Brill, J. P. 1988. Prediction of dispersed viscosity of oil/water mixture flow in horizontal pipes. SPE 18221. Presented at the *SPE Annual Technical Conference*, Houston, U.S.A.

McKibben, M. J., Gillies, R. G. and Shook, C. A. 2000. A laboratory investigation of horizontal well heavy oil-water flows. *Canada Journal of Chemical Engineering*. 78, 743-751.

McNulty, J. G. and Sutjipto, J. W. 1992. An investigation of transient two-phase flow patterns in a horizontal pipeline, SPE7040, *Annual Offshore Technology Conference*, Houston, Texas, USA, 4–7, pp.1–10.

Mimouni, S., Fleau, S. and Vincent, S. 2017. CFD calculations of flow pattern maps and LES of multiphase flows. *Nuclear Engineering and Design*. 321, 118–131.

Morgan, R. G., Markides, C. N., Zadrazil, I. and Hewitt, G.F. 2013. Characteristics of horizontal liquid-liquid flows in a circular pipe using simultaneous high-speed laser-induced fluorescence and particle velocimetry. *Int. J. Multiphase Flow*. 49, 99-118.

Oddie, G., Shi, H., Durlofsky, L. J., Aziz, K., Pfeffer, B., Holmes, J. A., 2003.Experimental study of two and three phase flows in large diameter inclined pipes. *Int. J. Multiphase Flow*. 29, 527–558.

Okufi, S., De Ortiz, E. S. and Sawistowski, H. 1990. Scale-up of liquid-liquid dispersions in stirred tanks. *Can. J. Chem. Eng*. 68(3), 400-406.

Parshall, J. 2012. Mature fields hold big expansion opportunity. SPE-1012-0052-JPT. *J Petroleum Technology*. 64 (10), 52 – 58.

Pei, P., Laudal, D., Nasah, J., Johnson, S. and Ling, K. 2015. Utilization of aquifer storage in flare gas reduction. *Journal of Natural Gas Science and Engineering*. 27 (2), 1100-1108.

Piroozian, A., Hemmati, M., Ismail, I., Manan, M. A., Rashidi, M. M. and Mohsin, R. 2017. An experimental study of flow patterns pertinent to waxy crude oil-water two-phase flows. *Journal of Chemical Engineering Science*. 164 (8), 313-332.

Poesio, P., Strazza, D. and Sotgia, G. 2008. Very-viscous-oil/water/air flow through horizontal pipes: pressure drop measurements and prediction, *J. Chem. Eng. Sci*. 64, 1136-1142.

Rafiee, S. E., & Rahimi, M. 2013. Experimental study and three-dimensional (3D) computational fluid dynamics (CFD) analysis on the effect of the convergence ratio, pressure inlet and number of nozzle intake on vortex tube performance–Validation and CFD optimization. *Energy*, *63*, 195-204.

Richardson, L. F. and Chapman, S. 1965. *Weather prediction by numerical process*. Dover Publications.

Rodriguez, O. M. H. and Oliemans, R. V. A. 2005. Experimental study on oil-water flow in horizontal and slightly inclined pipes. *Int. J. Multiphase Flow*. 29, 527-558.

Rozenblit, R., Simjhis, M., Hetsroni, G., Barnea, D. and Taitel, Y. 2000. Heat transfer in horizontal solid-liquid pipe flow. *Int. J. Multiphase Flow.* 26(8), 1235-1246.

Ruder, Z. and Hanratty, T. J. 1990. A definition of gas-liquid plug flow in horizontal pipes. *Int. J. Multiphase Flow*. 16(2), 233-242.

Ruspini, L. C., Marcel, C. P. and Clausse, A. 2014. Two-phase flow instabilities: A review. *Int. J. Heat and Mass Transfer*. 71, 521-548.

Russel, T. W. F., Hodgson, G. W. and Govier, G. W. 1959. Horizontal pipeline flow of mixture of oil and water. *Can. J. Chem. Eng*. 37, 9-17.

Russel, T. W. F. and Charles, M. E. 1959. The effect of the less viscous liquid in the laminar flow of two immiscible liquids. *Can. J. Chem. Eng*. 37, 18-34.

Serizawa, A., Feng, Z. P. and Kawara, Z. 2002. Two-phase flow in microchannels. *Experimental Thermal and Fluid Science*. 24(6-7), 703-714.

Shi, H., Jepson, W. P. and Rhyne, L. D. 2003. Segregated modelling of oil–water flows, SPE 84232. *SPE Annual Technical Conference*, Colorado, USA, 5–9, pp.1–10.

Shi, J., Gourma, M. and Yeung, H. 2017. CFD simulation of horizontal oil-water flow with matched density and medium viscosity ratio in different flow regimes. *Journal of Petroleum Science and Engineering*. 151, 373–383.

Soleimani, A., Lawrence, C. J. and Hewitt, G. F. 2000. Spatial distribution of oil and water in horizontal pipe flow. SPE66906, *SPE Annual Technical Conference and Exhibition*, Houston, U.S.A.

Sridhar, S., Zhang, H. Q., Sarica, C. and Pereyra. E. J. 2011. Experiments and model assessment on high-viscosity oil/water inclined pipe flows. SPE-146448-MS. *SPE Annual Technical Conference and Exhibition*, Colorado, U.S.A.

Strazza, D., Grassia, B., Demoria, M., Ferrari, V. and Poesio, P. 2011. Core-annular flow in horizontal and slightly inclined pipes: Existence, pressure drops and hold-up. *Journal of Chemical Engineering Science*. 66, 2853-2863.

SunderRaj, T., Chakrabarti, D. P., Das, G., 2005. Liquid–liquid stratified flow through horizontal conduit. *J. Chem. Eng. Technol.* 28(8), 899-907.

Surana, K. A., Allu, S., Tenpas, P.W. and Reddy, J. N. 2007. k-version of finite element method in gas dynamics: higher-order global differentiability numerical solutions. *International Journal for Numerical Methods in Engineering*. 69 (6), 1109–1157.

Taha, T. and Cui, Z. F. 2006a. CFD modelling of slug flow inside square capillaries, *Chemical Engineering Science* 61 (2), 665-675.

Taha, T., and Cui, Z. F., 2006b, CFD modelling of slug flow in vertical tubes. *Chemical Engineering Science*. 61 (2), 676-687.

Trallero, J. L., Sarica, C., Brill, J. P., 1997. A study of oil/water flow patterns in horizontal pipes. SPE36609, *J. SPE Produ. Facil.* 12, 165–167.

Yu, T. T., Zhang, H. Q., Li, M. X. and Sarica, C. 2010. A mechanistic model for gas/liquid flow in upward vertical annuli (SPE124181), *J. SPE Prod. Oper.* 25, 285-10.

Yuan, H. and Zhou, D. 2009. Evaluation of two phase flow correlations and mechanistic models for pipelines at horizontal and inclined upward flow, SPE120281, *SPE Production and Operations Symposium*, Oklahoma, USA, 4–8, 1–9.

Valle, A. 2000. *Three phase gas-oil-water pipe flow*. Ph.D. Thesis. Imperial College London.

van Baten, J. M. and Krishna, R. 2004. CFD simulations of mass transfer from Taylor Bubbles rising in circular capillaries. *Chemical Engineering Science*. 59 (12), 2535– 2545.

Vielma, M., Atmaca, S., Sarica, S. and Zhang. H. Q. 2007. Characterization of oil/water flow in horizontal and vertical pipes. SPE 109591. *SPE Annual Technical Conference and Exhibition*. Anaheim, U.S.A.

Vuong, D. H., Zhang, H. Q. and Li. M. 2009. Experimental study on high viscosity oil/water flows in horizontal and vertical pipes. SPE-124542-MS-P. *SPE Annual Technical Conference and Exhibition*, New Orleans, L.A., U.S.A.

Wallis, G. B., 1969. One-dimensional two-phase flow. *AIChe Journal*. 16, 896-1105.

Wang, Z. J., Fidkowski, K., Abgrall, R., Bassi, F., Caraeni, D., Cary, A., ... & Kroll, N. (2013). High-order CFD methods: current status and perspective. *International Journal for Numerical Methods in Fluids*, *72*(8), 811-845.

White, F. M. 1974. Viscous fluid flow. (3rd Ed.). U.S.A. McGraw-Hill.

Zhang, H. Q. Vuong, D. H. and Sarica, C. 2010. Modelling high viscosity oil/water flows in horizontal and vertical pipes. SPE-135099-MS-P, *SPE Annual Technical Conference and Exhibition*, Florence, Italy.

Zhou, Z. Y., Zhu, H. P., Wright, B., Yu, A. B. and Zulli, P. 2011. Gas-solid flow in an ironmaking blast furnace-II: Discrete particle simulation. *Powder Technology*. 208(1), 72-85.