COMPUTATIONAL FLUID DYNAMICS STUDY ON T-JUNCTION SEPARATOR FOR LIQUID-GAS SEPARATION

DINESH BALAKRISHNAN

PETROLEUM ENGINEERING

UNIVERISTI TEKNOLOGI PETRONAS

MAY 2013

CERTIFICATION OF APPROVAL

Computational Fluid Dynamics Study on T-junction Separator For Liquid-Gas Separation

by

Dinesh Balakrishnan

A project dissertation submitted to the
Petroleum Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(PETROLEUM ENGINEERING)

Approved by.

____
(Ms. Raja Rajeswary)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
MAY 2013

CERTIFICATION OF OR1GINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and the original work contained herein have not been undertaken or done by unspecified source or person.

DINESH BALAKRISHNAN

ABSTRACT

The separation of liquid-gas flow is an essential part of many industrial processes and the occurrence of multiphase flow in the petroleum industry is very common in the production and processing facilities of hydrocarbon. These separations are performed in large separator vessels under the effect of gravity containing large inventories of potentially flammable and/or toxic materials. The application of a simple defined partial phase separator (T-junction) would produce two streams, one rich in gas and the other rich in liquid. This would be beneficial to the petroleum industry especially in offshore oil platforms where safety, space, weight and cost are highly emphasized. Therefore, this project presents the study of fluid model on liquid-gas flow separation at a horizontal T-junction. Simulations were analyzed using numerical computational fluid dynamics (CFD) software. Numerical results of pressure profile and volume fraction of phases for fluid models are presented and analyzed to propose a new or improved T-junction design with better separation efficiency.

ACKNOWLEDGMENT

First of all, I would like to express my gratitude to my supervisor Ms. Raja Rajeswary. Her insight and passion aided me in the pursuit of new ideas and in the preparation of this project.

My appreciation also goes to my internship company supervisor, Dr Maung Maung Myp Thant @ Feroz Sultan and my project colleagues, Dr. Taufik and Mr. Faizal at PETRONAS GTS, without their expertise, training and guidance on Computational Fluid Dynamics (CFD) software, my conceptual ideas could never have been transformed into successful working realities.

My gratitude also goes to my colleagues, both past and present, for making a memorable postgraduate experience. I want to make thank to my course mate Zaihasfikrie, who has been an invaluable help throughout this process.

Finally, I reserve the biggest thanks for my family who have supported me throughout this adventure, especially my father and mother. They have demonstrated immense patience, enduring through the completion of this work, and continue to be an unending source of love and support.

God Bless

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1 BACKGROUND OF STUDY

In many industries, pipelines are known to be a medium used to transport fluid and studying its flow characteristics have been a challenge for many authors especially when dealing with multi-phase fluids and different pipe geometries. Multiphase fluid flow separation is achievable through T-junction pipes and such separations are desired to reduce problems when handling two or more phase mixtures as transportation of single phase streams are both safer and easier.

In the petroleum industry application, phase separation in T-junctions has been observed as early as 1973 by Orange. According to the field workers, the effect of two-phase liquid/gas separation at a pipe junction is almost unavoidable, having produced a stream of rich in liquid and another stream rich in gas. This has both negative and positive consequences. Among the negative consequences was described by Amir (Universiti Malaya, Malaysia). He suggests that in gas distribution networks, condensate can be formed in pipelines in winter due to low temperature and this condensate appears at some delivery stations while the other stations receive dry gas. This kind of uneven splitting may result in operational and separation problems. In the beginning, much research was attempted to minimise the phase redistribution problem. Over time, it soon became apparent that the same phenomena could be utilised in a positive way as partial phase separation. This positive remark was explored by Azzopardi et al. (2000), as they designed a T-junction incorporating a pipe work partial separator to replace a conventional vessel separator. A complete overview of the possible applications of T-junction as partial separator can be found (Robert, 1994; Rea, 1998; Azzopardi, 1999).

Though there is by now a reasonable experimental study on the performance of such junctions, the data obtained will be based on air/water at near atmosphere pressure in small diameter which does not exactly simulate the actual phase split at the junction, (Baker 2003). Moreover, Azzopardi (1999) cited that there is yet to be an agreed criterion to identify at what conditions and for with what geometry a T-junction is a good (partial) phase separator.

1.2 PROBLEM STATEMENT

The maldistribution (unequal distribution) of phases by a junction can be a major problem in downstream equipments (Mak et al., 2006). This statement is supported by Azzopardi *et al.* (2000) stating that maldistribution can result in a fall of efficiency in downstream equipment. He cited example of phase maldistribution that has been reported from offshore platforms in the UK North Sea.

"A two main phase vessel separators are decided to be installed in parallel. This would enable production to continue even at a reduced rate if there was a need for maintenance or modification of a separator. To ensure even split of the fluid phase, an impacting T-junction with both outlet pipes were at a right angle to the inlet. When the system was started up, it was found that one of the separator received most of the gas while the other got most of the liquid. Inspection has showed that the outlet angle was centrifuging the phases and presenting each outlet with substantially one phase."

According to sources from PETRONAS, most un-manned platforms use produced gas as fuel to generate power to run its facilities and control system. This produced gas is tapped directly from gas lift or production header (Figure 1). In the certain situations, the gas scrubber is unable to isolate the liquid and gas due to the large amount of wet gas channelled into it. As result, many platforms experience problems in the instrument gas systems causing frequent trips and maintenance. Once a platform trip, it can take up to two days to restart and stabilise oil production and for platforms which have gas lifted wells, it can take up to 2 weeks to reach and stabilize oil to target rates. Therefore, the correctly design T-junction is crucial to achieve the highest phase separation efficiency to avoid the aforementioned problems and their associated issues.

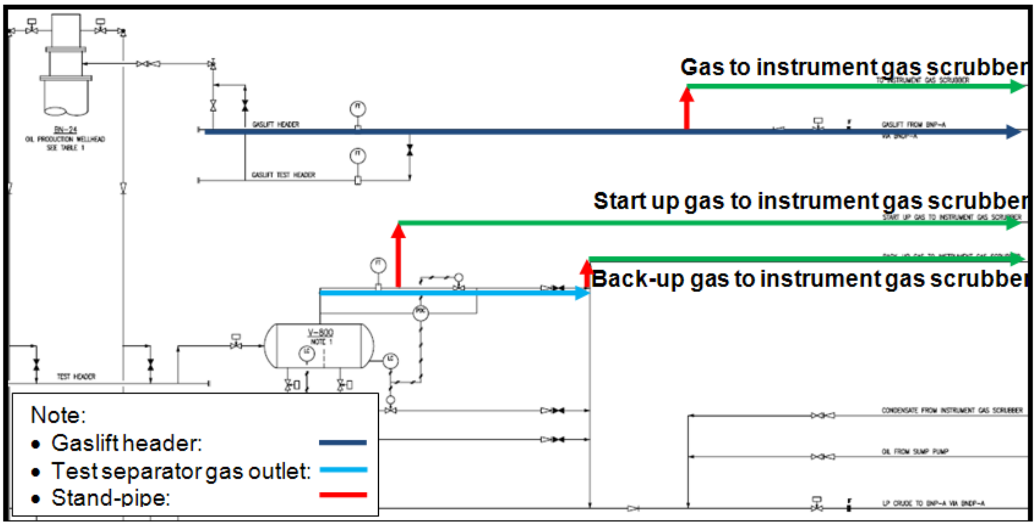


Figure 1: Schematics of an instrument gas system

1.3 OBJECTIVES

After highlighted the requirement of liquid-gas separation in a T-junction acting as a partial separator, few important objectives of this work are quantified.

The first objective is to apply knowledge of the flow split of liquid-gas flows and T-junction to develop a novel/improved partial phase separator. This could be based on an alteration to the T-junction geometry design or control in fluid flow parameters.

The second objective is to apply numerical study to simulate the separation of liquid and gas including petroleum fluid flow through the T-junction and to analyze the separation efficiency of the T-junction designs. This study is achieved by performing Computational Fluid Dynamics (CFD) study on the T-junction.

The third objective is to perform a fluid model validation of the simulation work with other published experimental or simulation results by different authors. Lahey (1986) in his review, concluded that "no completely satisfactory model exist for the prediction of phase separation in conduits of untested geometry and operating conditions."The validation parameters can vary from pressure drop, volume fraction and etc., depending on the parameter availability. For some cases, the fluid model will be validation between theoretical result and obtained simulation result. Once a fluid model is valid, the geometry of the T-junction will be altered to increase the separation efficiency.

1.4 SCOPE OF STUDY

Understanding the flow behaviour of a two-phase flow is much more complex than the single-phase flow as two-phase flow involves processes with many variables. Therefore, the scope of study covered for this work is the liquid-gas flow in horizontal T-junction configurations. A good understanding of any liquid-gas system is crucial for the design of plant equipment. One key issue is the understanding and prediction of the two-phase flow regime present during the simulation process as a method of fluid model validation with the experiment data.

From the design aspect of the T-junction, many different geometry of T-junction is used for experiment and simulation purposes. For this work, the emphasis will be on a diverging T-junction which consists of 1 inlet and 2 outlet zones. The T-junction design of study will be a horizontal main run with a 90^o vertical pipe (with no internal attachment) fitted/welded to it as shown in Figure 2. Simulations will be conducted using this design to study the separation pattern and subsequently alter the design to study its separation performance.

Finally, determining the correct simulation fluid model is necessary to validate the simulation result against experiment or simulation results by other authors. This is the peak of the process as after model validation, re-configuring of T-junction geometry can be done to perform sensitivity study on separation efficiency.

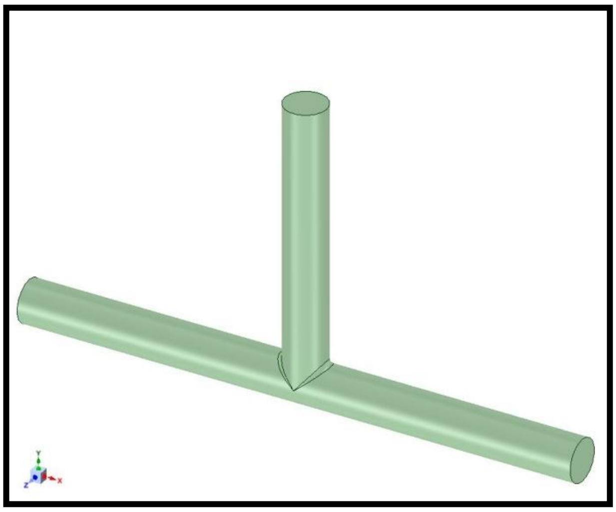


Figure 2: Isometric sketch of horizontal T-junction with +90 degree vertical branch

CHAPTER 2
LITERATURE REVIEW

2.1 MULTIPHASE FLOW

According to Christopher (2005), the term multiphase flow refers to any fluid flow consisting of more than one phase (solid/liquid/gas) or components. The emphasis of this work particularly is on two-phase (liquid-gas) fluid flow in a horizontal pipeline. Understanding the flow behaviour of a two-phase flow in much more complex than the single-phase flow as two-phase flow involves processes with many variables. Gas and liquid phases do not flow at the same velocity in a pipeline because of the difference in their densities and viscosities (Ottens *et al*., 1999). These variables are important factors to distinguish two-phase flow from the normal single-phase flow. The flow pattern in multi-phase flow can vary from horizontal to vertical flow following the geometry of the pipeline. Taitel and Dukler (1976) cited that the different flow pattern are formed because forces that act on the fluids, such as bouyancy, turbulance, inertia, and surface tension. All these fluid forces vary with flow rates, pipe diameter, inclination angle, and fluid properties of the phases. Rouhani and Sohel (1983) cited a survey which suggested 84 different flow patterns. For some cases, names are given to flows which have common geometric flow pattern. The four major flow patterns exist in a horizontal pipe are illustrated in Figure 3 below.

In the case of horizontal co-current liquid-gas flow, gravity force acts perpendicular to the direction of motion (Baker, 2003). There will be a distinct liquid-gas boundary since gravity will have a much larger effect on the denser liquid phase. He defined the following parameters among horizontal flow.

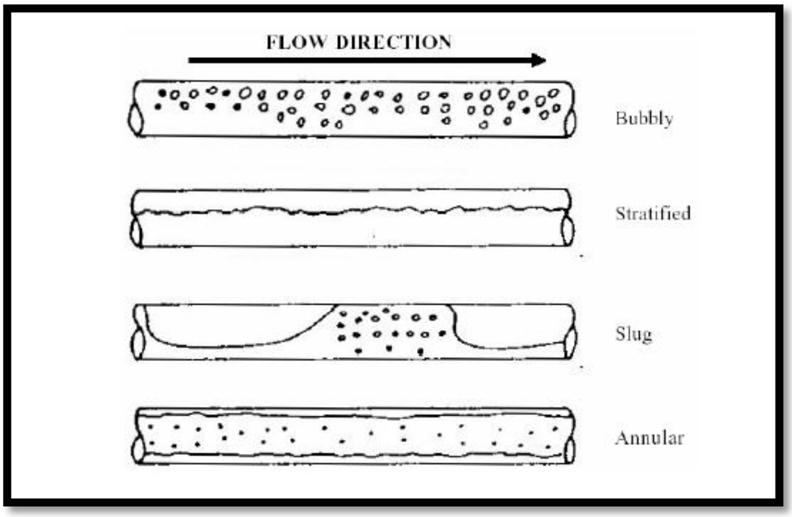


Figure 3: Two-phase flow pattern in horizontal pipe (Beggs and Brill, 1973)

Bubbly Flow:

Non-uniformed size gas bubbles are distributed within the liquid phase. Bubbles tend to accumulate at the top of the pipe due to gravity, except at very high liquid velocities where bubbles disperses around the entire pipe cross-section.

Stratified Flow:

There is a clear and smooth interface of continuous liquid layer flowing along the lower section of the pipe with the gas flowing about it when the superficial velocity of gas is low. As the gas superficial velocity increases, waves are formed creating stratified-wavy flows.

Slug Flow:

The increase liquid superficial velocity causes waves in stratifies-wavy regime become large enough to lead to the intermittent flow pattern termed slug flow, where the gas phase will travel in large pockets at the top of the pipe.

Annular Flow:

At very high gas velocities, some of the liquid if forced around the wall of the pipe while the rest travels as entrained droplets within the central gas flow. Gravity tends to force the liquid to the bottom of the pipe making liquid film thicker.

2.2 T-JUNCTION

T-junction is a very common component in pipe networks, mainly used to distribute (diverge) the floe from main pipe to branching pipe or to accumulate (converge) flows from many pipes to a single main pipe (Mohammed Abdulwahhab et al., 2012). In this work, the attention is giving to diverging T-junction which consists of 1 inlet zone and 2 outlet zones, as shown in Figure 4.

There will be eight variables required to fully define the two-phase flow in a T-junction (Bakers, 2003). These are the mass fluxes in each arm, $\dot{M}_1$, $\dot{M}_2$, $\dot{M}_3$, the quality of these streams, x1, x2 and x3, and the associated pressure drops, ΔP_{12} and ΔP_{13}. The suffixes 1, 2, and 3 indicate the inlet, run and branch arm, respectively as shown in Figure 2.

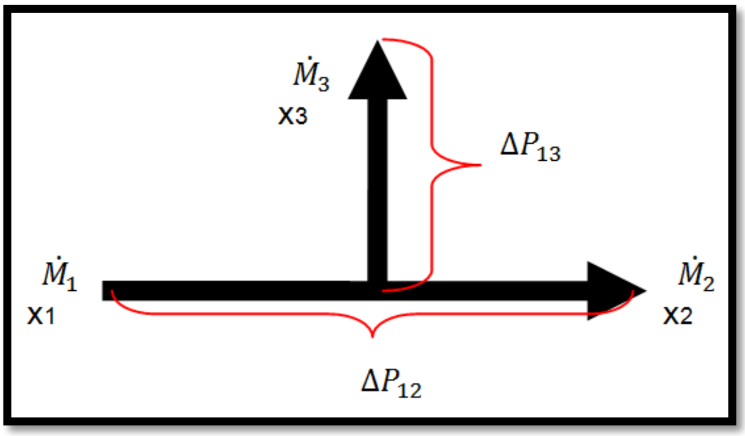


Figure 4: Parameters involved in T-junction problem

Wren (2001) in this work cited 3 dominant forces effecting two-phase flow separation and understanding of the dominant forces will help us understand better the phase split at a horizontal T-junction. These forces are considered to be:

- **Gravity:** Gravitational acceleration will act mainly on the liquid phase due to its higher density and this encourages liquid phase to settle at the bottom of the pipe, minimizing the liquid taken off when the side-arm is angled upwards (Baker, 2003).

- **Inertia:** The liquid phase will travel along the pipe with a much higher momentum than the gas due to its relatively higher mass, forcing liquid to continue along the pipe, bypassing the entrance to the side-arm. If the side-arm has a reduced diameter, the liquid will have even less time creating a better separation for gas due to be gravity influence (Azzopardi & Whalley, 1982; Azzopardi, 1984; Charron & Whalley, 1995).

- **Pressure:** Walter et al. (1998) discovered that the pressure drop exists between the inlet and side-arm and a pressure recovery into the run. This recovery is similar to Bernoulli Effect for single phase flow, produced as a result of the decrease in the mixture velocity in the run. Figure 5 shows a typical pressure profile for a junction.

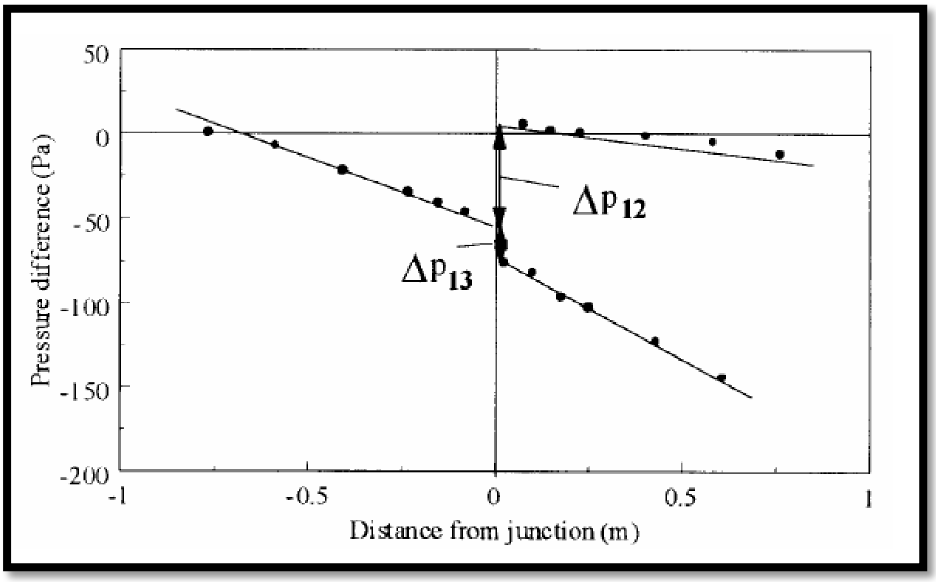


Figure 5: Pressure drop profile across a T-junction (Baker, 2003)

2.3 TWO-PHASE FLOW PROPERTIES

2.3.1 Fundamental Definitions

This section introduces the primary variables used throughout this work. To distinguish between gas and liquid, the subscripts 'L' for liquid and 'G' for vapour will be used. Two-phase flow is the simplest case of multiphase flow in which two phases are present for a pure component (Moreno, 2005). Moreno's analytical and experiment studies have presented the following variable definitions.

Vapor Quality

The vapour quality (χ) is defines to be the ratio of the vapour mass flow rate ($\dot{M}_G$ $[kgs^{-1}]$) divided by the total mass flow rate($\dot{M}_G + \dot{M}_L$):

$$x = \frac{\dot{M}_G}{\dot{M}_G + \dot{M}_L}$$

When phase change does not take place in a channel, the mass flow rate of each phase is measured, and the quality is then determined for the entire channel.

Void Fraction

Void fraction is one of the most important parameters to be defined in two-phase flow. If defines the cross-sectional area occupied by each phase. As it determines the mean velocities of the liquid and vapour, it represents a fundamental parameter in the calculation of pressure drop, flow pattern transitions and heat transfer coefficients.

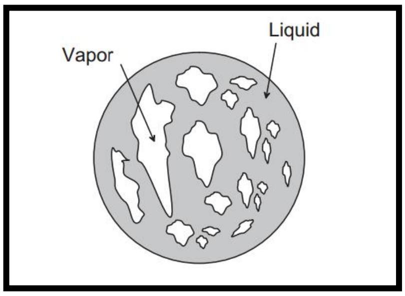


Figure 6: Cross-sectional void fraction

The void fraction of the vapour is defined as:

$$\epsilon = \frac{A_G}{A_G + A_L}$$

Where A_G is the sum of areas occupied by the voids and A_L is the sum of areas occupied by the liquid. The total cross-sectional area of the channel is called A.

Velocity

In two-phase flow, there are 2 different velocities that can be defines. In general, the phases will not have the same velocity and there will be a relative velocity between them. The first type is the true average velocities or also known as actual velocity of phase's υ_G and υ_L are the velocities which the phases actually travel. The cross-sectional average true velocities are determined by the volumetric flow rates $\dot{Q}_G and\ \dot{Q}_L$ $[m^3 s^{-1}]$of the vapour and liquid divided by the cross-sectional areas occupied by the respective phases:

$$u_G = \frac{\dot{Q}_G}{A_G} = \frac{\dot{Q}_G}{\epsilon A}$$

$$u_L = \frac{\dot{Q}_L}{A_L} = \frac{\dot{Q}_L}{(1-\epsilon)A}$$

The second type of velocity is the superficial velocities also known as volumetric fluxes of the phase's j_G and j_L are defined as volumetric flow rate of the phases through the total cross-sectional are of the two-phase flow. It might be expressed as the phase velocity if it would flow alone in the entire cross section.

$$j_G = \frac{\dot{Q}_G}{A} = \frac{G}{\rho_G} x = \epsilon\, u_G$$

$$j_L = \frac{\dot{Q}_L}{A} = \frac{G}{\rho_L}(1-x) = (1-\epsilon)\, u_L$$

2.3.2 Flow Pattern Identification

In many industries, there is a requirement not just to understand the possible flow patterns but to also predict which flow regime exists within a given pipeline (Baker, 2003). Two-phase liquid-gas flows can form various configurations as it travels along a horizontal pipe, as outlined in Chapter 2.1 of this report, all of which will have different characteristics. To quantify the different flow regimes that may be expected within a pipe, the concept of using flow pattern maps was developed.

Despite many attempts by researchers with the use of high-speed video photography, visual inspections of fluid flow pattern is still considered more appropriate and accurate. In industrial situations, where the pipelines are not transparent, more instrumental-based techniques are required. Barnea and Taitel (1985) outlined several methods for measuring void fractions or pressure fluctuations in two-phase flows. To detect the flow patterns within a pipe network, a two-dimensional plot were developed to display transition boundaries of the different fluid. There are two basic types of coordinates used for mapping; one uses dimensional axes (e.g. superficial velocities, mass flow rates) while the other utilizes dimensionless groups (e.g. Froude number, Reynolds number, gas-liquid mass ratios).

Over many years of research and experimental work, there have been many different horizontal flow pattern maps suggested for various flow conditions. One of the earliest maps was created by Baker (1954). This map is still popular within the petroleum industry due to its simplicity of function. However, further work by the author shows that the transitions boundaries between each flow regime cannot be predicted easily even by visual observation. Mandhane *et al.* (1974) noticed that the superficial velocities of the gas and liquid phases are the major influence on the flow pattern and mapped a significant data by coordinated system of superficial gas velocity and superficial liquid velocity, locating the transition lines.

Later, Taitel and Dukler (1976) modified Mandhane *et al*. (1974) idea and produced a flow map based on the mechanisms of flow regime transitions. These transitions are between five basic flow regimes; stratified smooth, stratified wavy, intermittent, annular and bubbly. His analysis began with stratified smooth flow as the initial flow pattern that would occur in low superficial fluid velocity and examined the change in regime with the increase of superficial velocity. Figure 7 shows a typical flow pattern map produced using the methodology of Taitel and Dukler (1976).

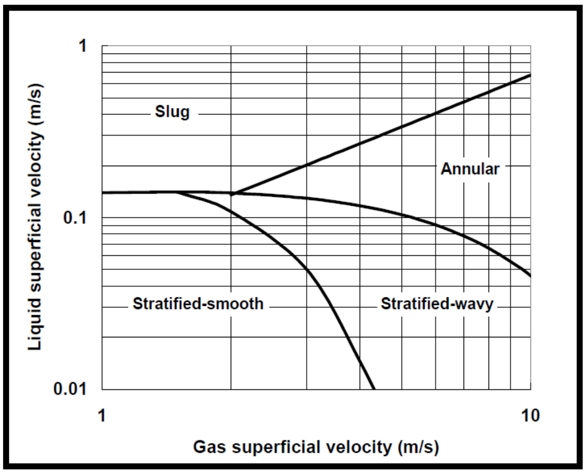


Figure 7**:** Typical horizontal flow pattern map based on methodology of Taitel and Dukler (1976)

2.4 EXPERIMENTAL MODELS

For the past 25 years, many experiments were conducted to study the phase split of a two-phase flow in a T-junction. These experiments vary from types of fluids used, fluid velocity, T-junction orientations and some with modified T-junctions for different flow regimes. Most of the result is presented in terms of liquid-gas fraction take-off from the main. However, in this report, our emphasis will be on experiments with diverging horizontal T-junctions with vertical upwards side-arm. The following Table 1 is a summary of experiment settings from sources which coincide with our work emphasis.

Table 1: Previous works with horizontal and vertical upwards T-junction

Author(s) and year	**Fluid**	**Main pipe orientation**	**Branch orientation**	**Main pipe diameter (m)**	**Diameter ratio**	**Flow patterns studied**
Hong (1978)	Air/water	Horizontal	0, ±(45, 90)	0.0095	1.0	Annular Stratified
Azzopardi & Whalley (1982)	Air/water	Vertical & Horizontal	0, ±(30, 60, 90)	0.032	0.40	Annular
Katsaounis (1987)	Air/water	Horizontal	+90	0.203	0.40	Plug Stratified
Reimann *et al.* (1988)	Air/water & steam/water	Horizontal	0, ±90	0.05	1.0 0.52	Stratified
Mudde *et al.* (1993)	Air/water	Horizontal	+90	0.23	0.43	Stratified

All the authors mentioned in the above table, agree on a same result, an even split of liquid-gas was obtained in the vertical upward side branch for stratified and annular flow. Whilst the majority of experimentalists have worked with small diameter pipe work, Maciaszek and Micaelli (1988) and Mudde et al. (1993), performed experiments on more industrial sized equipments. Despite their large diameter pipe work, result shows a reasonable split due to the low fluid velocity at the T-junction giving enough time for liquid to flow passing through the branch opening with its gravity and inertia force.

2.5 SIMULATION MODELS

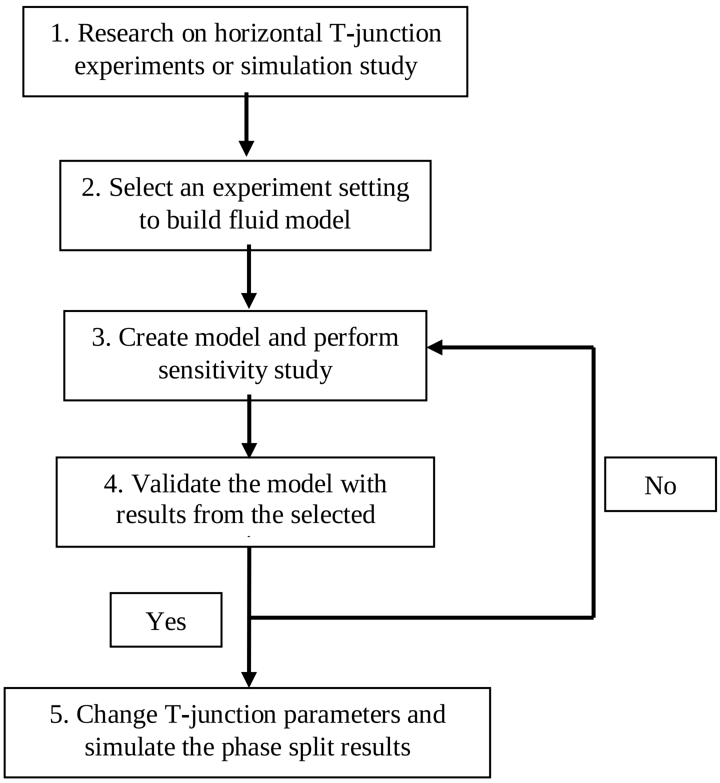
Compared to experimental works, simulation studies on T-junction less performed and has more uncertainties on the simulation fluid model. To validate a simulation model, a simulation result must be verified with experimental result. The advantage of simulation is that once a model is validated, parameters around the model can be restructured to predict the possible outcomes. Adechy and Issa (1999) and Adechy (2000) developed a CFD simulation model for annular flow model and compared it with present experimental works for validation purposes. Azzopardi and Whalley (1982) and Robert *et al.* (1997) also developed a model to predict the split of the phases. Compared to two-phase flow, single phase flow simulations are easier and the fluid models are much more accurate as presented by Mohammed *et al.* (2012). Till date, there is no CFD model developed in accordance to the emphasis of this project. Thus, the simulation model needs to be freshly developed and validated later with previous verified experimental results. The nearest CFD fluid model in a horizontal T-junction available is a 3 phase flow of Petroleum, Gas and Water in T-junctions by Cavalcanti *et al*. (2011).

CHAPTER 3
METHODOLOGY

The following methodology is a guideline system for solving the project problem by obeying the objectives mentioned earlier in this report, with specific components such as project activities, key milestones, Gantt-chart and tools.

3.1 PROJECT ACTIVITIES

To visualize the project activities, a flow diagram (Figure 8), is created. The main aim of this 7 step process is to obtain the accurate, verified and usable fluid model for the analysis of T-junction. Detailed explanation of every process will be discussed further is this report.



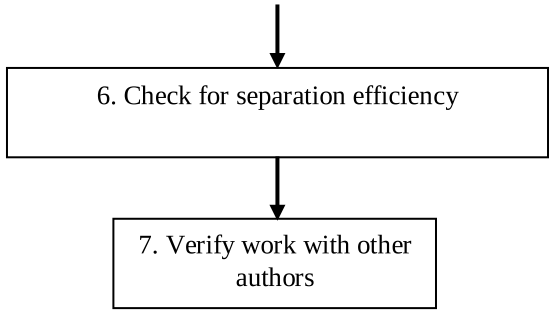


Figure 8: Project activity flow diagram

Step 1 is the literature research that based on the emphasis of our project work, which two-phase liquid-gas flow through a horizontal T-junction. This literature consists of experimental and simulation work as summarized in Chapter 2.4 and 2.5. Since the project work is simulation based, and a two-phase separation on a T-junction experiment and a two-phase oil-liquid-gas simulation study set-up is selected to be simulated.

At step 2, the experiment carried out by Azzopardi *et al.* (2000) is selected for simulated to create a fluid model. The experiment was conducted in at onshore oil field in the south of England. The experiment configuration had been design to specifically promote homogenous flow into the test section. Azzopardi did not carry out any simulation work to support the experiment results and this is a disadvantage because many assumptions (more sensitivity study) have to be made in order to get the right fluid model. Based on Figure 9, the junction of study is the 0.076m diameter vertical side arm T-junction. The following are the physical properties of the fluids as shown in Table 2.

Table 2: Physical properties of the fluids

Property	**Values**
Gas density (kg/m^3)	20-25
Liquid density (kg/m^3)	790
Gas viscosity (Pa s)	0.000013
Liquid viscosity (Pa s)	0.001
Gas superficial velocity (m/s)	0.76
Liquid superficial velocity (m/s)	0.74
Surface tension (N/m)	0.015

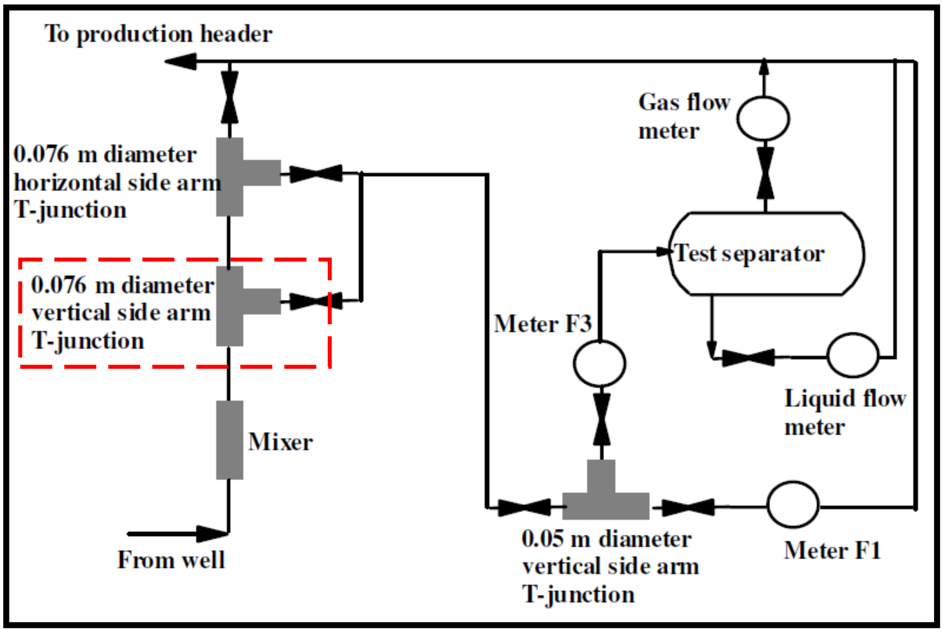


Figure 9: Schematic arrangement of test facility

Besides simulating an experiment work, a simulation study of 3 phase flow (liquid-liquid-gas) of Petroleum, Gas and Water in T-junction by Cavalcanti *et al*. (2011) is also simulated. Based on an expert opinion from PETRONAS, if a 3 phase flow model can be constructed, two-phase flow model can be reconfigured from it. For the simplification of the fluid model, Cavalcanti *et al*. proposed few assumptions: steady-state flow, laminar flow, three-dimensional flow, incompressible flow, non-isothermal flow, Newtonian fluid, and constant thermophysical properties. The geometric characteristic of the junction, physical properties of fluid, and the fluids characteristic in the inlet are tabulated in their respective Tables 3-5 below.

Table 3: Geometric characteristics of the junction

	Diameter		**Length**	
Type of junction	**Main branch (cm)**	**Side branch (cm)**	**Main branch (cm)**	**Side branch (cm)**
Horizontal T	10	9.5	100	50

Table 4: Physical properties of the phase used in the simulation

Physical Properties	Values	Sources
Water density (kg/m^3)	1000	*Incropera e DeWitt (2002)*
Water viscosity ($N.s/m^2$)	1.0×10^{-3}	*Incropera e DeWitt (2002)*
Gas density (kg/m^3)	1.12	*Rohsenow et al. (1998)*
Gas viscosity ($N.s/m^2$)	1.78×10^{-5}	*Rohsenow et al. (1998)*
Oil density (kg/m^3)	951	*Incropera e DeWitt (2002)*
Oil viscosity ($N.s/m^2$)	0.5	*Incropera e DeWitt (2002)*
Surface tension coefficient (N/m)	0.08	....

Table 5: Characteristics of the phases in the inlet of the junction

Phases	Velocity (m/s)	Volume fraction	Flux	Particle diameter (mm)
Oil	0.03	0.7	Continues	....
Water	0.03	0.2	Dispersed	8
Gas	0.03	0.1	Dispersed	1

Step 3 is to create a base fluid model based on the properties and data provided by the author and analyze it. Further modifications will be made from the base model and analyzed again to simulate the experiment and simulation result. This loop process is call sensitivity study. There are 7 fluid model create for the experiment data and 1 fluid model for the simulation data. The properties of all the fluid model is provided in Appendix A.

Step 4 is the validation of the created fluid models. For the experiment models, each model case will be compared and analyzed in terms of fraction of liquid and gas taken-off. The simulation model will be compared and validation with the simulation result provided by the author, which is in terms of mass flow rate (kg/s).

Step 5, 6 and 7 is the path of the main aim of this project. After a validated model is obtain, a novel T-junction design, horizontal inlet with vertical upward and downward outlets is proposed to study the fluid separation and for better separation efficiency analysis. The proposed T-junction is a regular (same diameter) design for the inlet and the two outlets as shown in Figure 10.

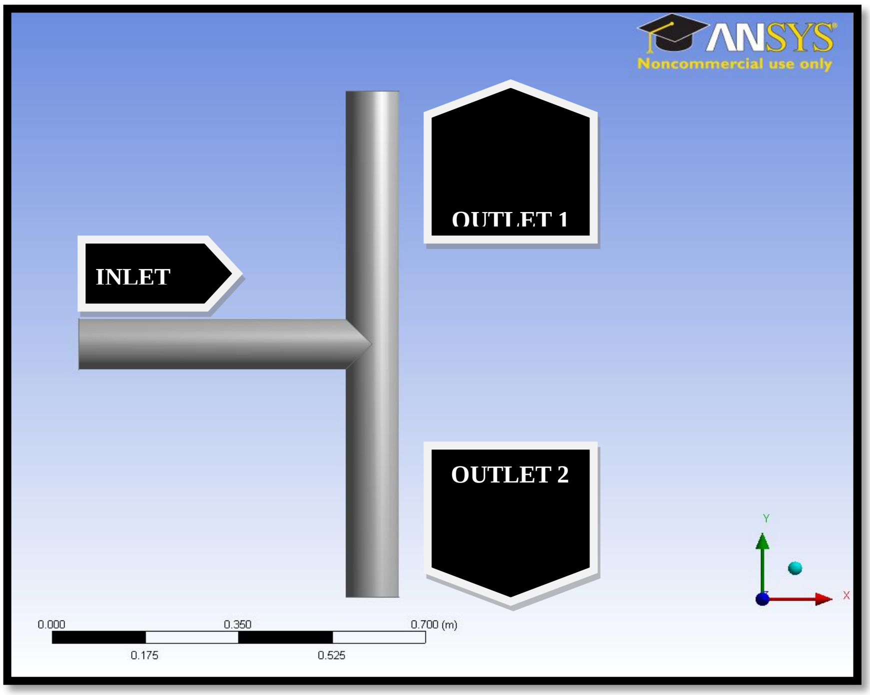


Figure 10: Proposed T-junction design with regular diameter openings

The geometric characteristics of the junction are obtained from a design created by Cavalcanti *et al*. (2011). The alteration of the vertical upward and downward branch was made after many researches and study on effect of orientation of side arm to phase separation.

Table 6: Geometric characteristics of the proposed T-junction design

	Diameter			**Length**		
Type of junction	**Main branch (cm)**	**Vertical upward branch (cm)**	**Vertical downward branch (cm)**	**Main branch (cm)**	**Vertical upward branch (cm)**	**Vertical downward branch (cm)**
Horizontal T	10	10	10	50	50	50

3.2 GANTT CHART & KEY MILESTONE

Table 7: Project Gantt Chart and Key Milestones

No.	Details / Weeks	1	2	3	4	5	6	7		8	9	10	11	12	13	14	15
1	**Project Work Continues** • Research on horizontal T-junction experiments or simulation study • Select an experiment setting to build fluid model • Create model and perform sensitivity study • Validate the model with results from the selected	■	■	■	■	■	■	■	MID SEM BREAK								
2	**Submission of Progress Report**							●									
3	**Project Work Continues** • Change T-junction parameters and simulate the phase split results • Check if the separation criterion and efficiency • Verify work with other authors									■	■	■	■	■			
4	**Pre-SEDEX**											●					
5	**Submission of draft Report**												●				
6	**Submission of Dissertation (soft bound)**													●			
7	**Submission of Technical Paper**													●			
8	**Oral Presentation**														●		
9	**Submission of Project Dissertation**																●

Legends:

■ **Progress** ● **Key Milestones**

3.3 TOOLS

3.3.1 CFD Method

CFD is a simulation method used to modify the design and to improve the operation of most types of chemical process equipment, combustion systems, flow measurement and control systems, material handling equipment and pollution control system (Shelley, 2007). In other words, CFD is the science of predicting fluid flow, heat and mass transfer, and related phenomena by solving mathematical equations using numerical methods and algorithms. The following flow chart shows the step-by-step procedures to perform CFD analysis.

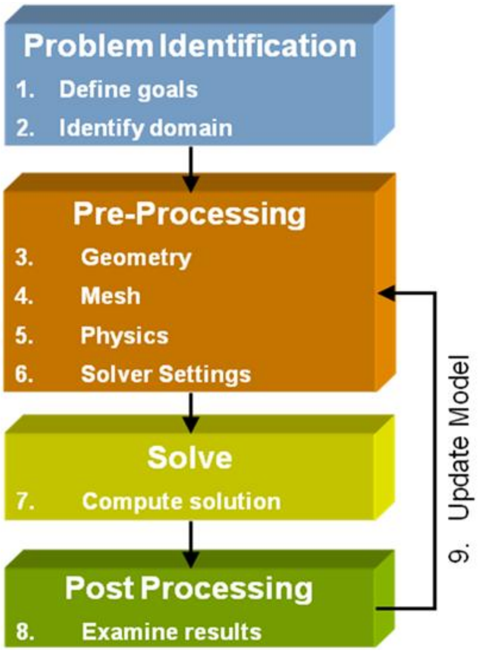


Figure 11: Sequence of Steps on the CFD analysis execution

At the problem identification stage, the goal of analysis (heat transfer, pressure profile, volume fraction or etc.) need to determined. This is followed by the identification of domain of the study. During the Pre-Processing stage, geometry of the domain is created using 3D software called ANSYS SPACECLAIM or ANSYS Design Modeller. The developed geometry will then be meshed (discretized into finite set of volumetric cells). Using ANSYS-FLUENT or CFX software, the fluid properties, flow models and convergence criteria are specified. The collected boundary and initial conditions are applied to the model.

All the data defined in the pre-processing stage are fed into the solver programme and solves the numerical equations based on the data specified. The results are written to a result file using post-processing software. Once a converged solution is obtained, the results are analyzed through variety of methods such as contour, plan, vector or line plots to check the satisfactory of the solution. If the result is unsatisfactory, the error needs to be identified. The step 3 to 8 of the CFD analysis is repeated several times with different types of model to choose the best flow model.

CHAPTER 4
RESULTS AND DISCUSSION

4.1 RESULTS

4.1.1 Simulation of Azzopardi *et al*. (2000) experiment work

As mentioned earlier in the project activities, there are 7 different fluid models created to achieve the similar result as obtained through the experiment work. The 7 fluid models are as shown in Table 8 below.

Table 8: Simulation Fluid Models

Case	Fluid Model
1	Base
2	Bouyant_Maximum Density
3	Bouyant_Mininum Density
4	Dispersed Gas
5	Free Surface
6	Free Surface and Bouyant_Maximum Density
7	Free Surface and Bouyant_ Mininum Density

Based on these fluid models, the fraction of liquid-gas taken-off is calculated for respective models, phase separation data tabulated and compared with experiment results as shown in the Figure 12 below. The experiment result is represented on an x-y axis graph of fraction of liquid taken-off versus fraction of gas taken-off.

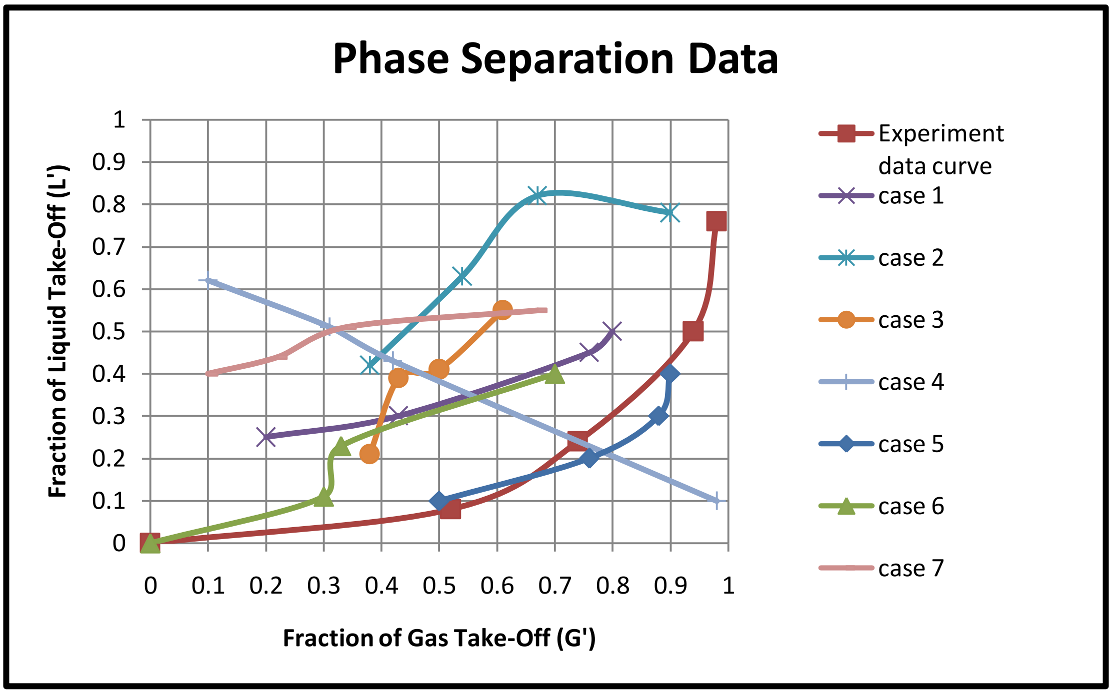


Figure 12: Phase separation data of fluid model and experiment result

From Figure 12, it can be observed that none of the cases match the experiment data curve. However, case 1 and case 5 have similar trend to the experiment data curve and data's of case 5 is significantly near to the data's of the experiment curve. For case 2, 3, 6 and 7, the curves are irregular and have no relationship to the experiment data curve. For case 4, the curve is not proportional to the experiment data curve. Case 5 seems to be fluid model that can be used for the study of the new proposed T-junction. To further validate case 5 fluid model, an error analysis is conducted to calculate the error obtained from the model as show in Table 9 below.

Table 9: Error analysis on Experiment data against Case 5 data

Experiment		**Simulation Case 5**		**Error (%)**	
Fraction of Gas Take-off (G')	**Fraction of Liquid Take-off (L')**	**Fraction of Gas Take-off (G')**	**Fraction of Liquid Take-off (L')**	**Fraction of Gas Take-off (G')**	**Fraction of Liquid Take-off (L')**
0.52	0.08	0.5	0.1	3.8	25
0.74	0.24	0.76	0.2	2.7	16
0.94	0.5	0.88	0.3	6.3	40
0.98	0.76	0.9	0.4	8.2	47

Based on Table 9, the higher percentage error is calculated from the fraction liquid taken-off. The highest percentage error is 47% for fraction of liquid taken-off (L') and the lowest percentage error is 2.7% for the fraction of gas taken-off (G').

4.1.2 Simulation of Cavalcanti *et al.* (2011) simulation study

In this simulation study, only 1 fluid model was created because most of the data and conditions are provided by the author in this report. Despite less assumptions made during the construction of the fluid model compared to the experimental work, the error analysis performed indicates that the current fluid model developed is not 100% accurate. The following is the numerical mesh comparison made between the author's simulation mesh and the current developed simulation mesh.

Table 10: Numerical mesh comparison

	Number of Elements	**Number of Nodes**
Cavalcanti' simulation	140,908	48,028
Current simulation	144,417	47,918
Error (%)	2.4	0.23

From Table 10, the number of elements for the current simulation is lower compared to Cavalcanti's meshing and the direct opposite for the number of nodes. Cavalcanti's meshing has produced a higher number of nodes compared to the current simulation meshing, provided a very small percentage of error.

After the meshing is refined to the nearest number of elements and nodes of the author's work as much as possible, simulation was set up and ran. The result, current simulation has produced the exact mass flow rate as presented by the Calvalcanti and is tabulated in Table 11 below.

Table 11: Mass flow rate at the outlets of the junction

	Mass flow rate (kg/s)		
	Water	**Gas**	**Oil**
Outlet 1	0.1845	0.0001154	0.11826
Outlet 2	0.05110	0.0001485	0.10584

Based on Table 11, the outlet 1 is specified as the horizontal outlet and outlet 2 is the vertical outlet. The mass flow rate of gas at outlet 2 is higher (56%) compared to the mass flow rate at outlet 1 (44%) as this is a logical scenario. Both water and oil flow rate is higher at outlet 1 compared to outlet 2, at 78% and 52% percent respectively. This result can further be supported with a graphical presentation of gas volume fraction in the T-junction as shown in Figure 13 below.

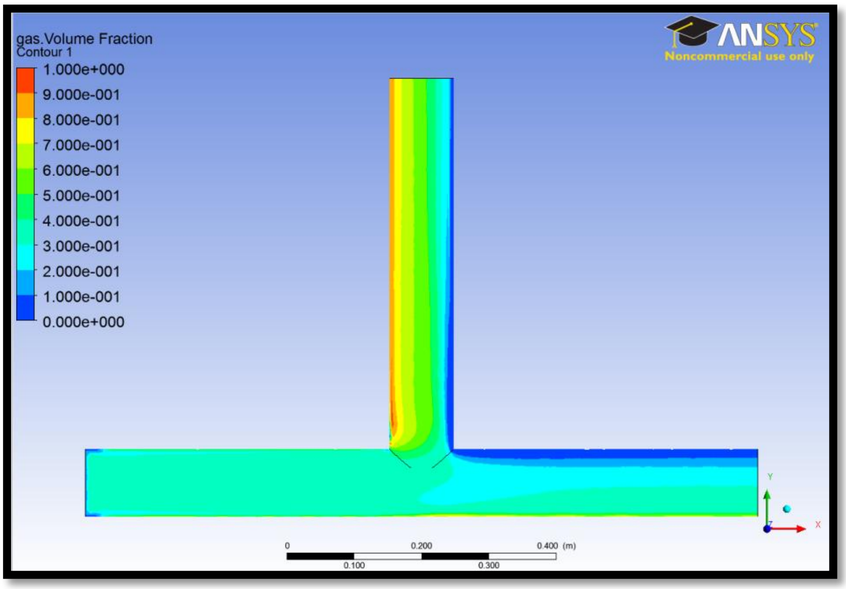


Figure 13: Gas volume fraction representation in T-junction

Based on Figure 13, there is a gas fraction concentrate at the outlet 2 compared to outlet 1 which explains the higher percentage of mass flow rate at outlet 2. Gas fraction is at 0.5 at the inlet as it mixes with the incoming fluid of oil and water. As the mixed fluid reaches the junction, gas propagates to the vertical upward side arm and the higher density fluid (water and oil) passes the outlet 2 opening and continues to outlet 1.

4.1.3 Simulation of proposed T-junction design with Cavalcanti's fluid model

As mentioned earlier in the project activities, the new design is proposed based on the research on the effect of geometry orientation on the phase split efficiency. The proposed design is meshed (Figure 14) with the same specifications used by Cavalcanti *et al.* (2011) in this simulation geometry meshing. These specifications are tabulated in Table 12 below.

Table 12: Meshing specifications for proposed T-junction design

Mesh Specification	Value
Number of elements	143, 622
Number of nodes	47, 240
Minimum mesh size	1.7087 E-0.04 meter
Maximum mesh face size	9.7 E-0.03 meter
Maximum mesh size	9.7 E-0.03 meter

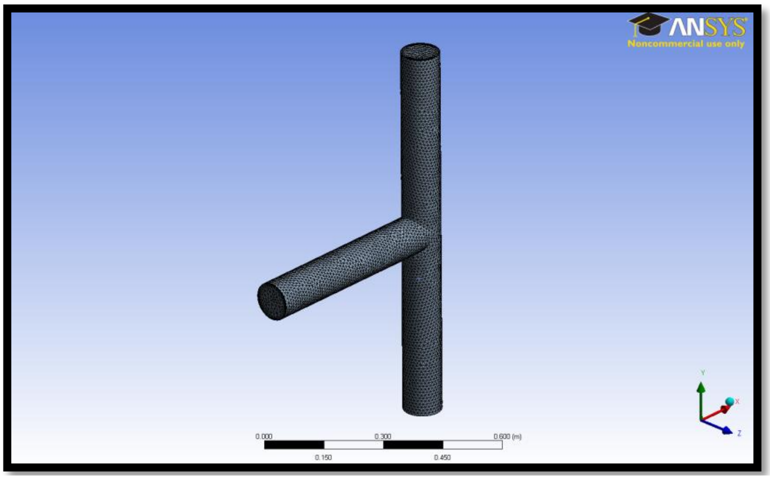


Figure 14: Details of meshing of proposed T-junction design

After the geometry created and meshed into discrete finite volume, the fluid model developed to simulate Cavalcanti's simulation result was used on the proposed design and the following results were obtained.

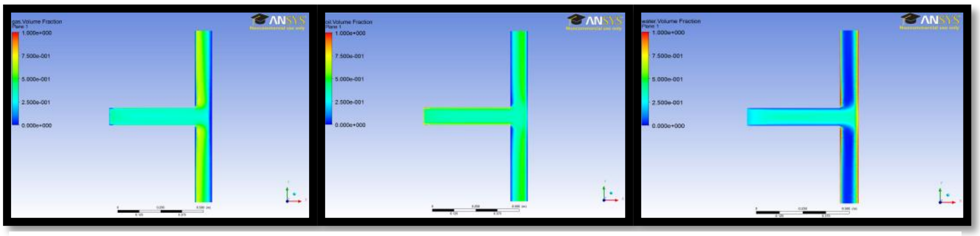


Figure 15: Volume fraction of gas, water and oil in the x-y plane of the proposed T-junction

Figure 15 shows the volume fraction of gas, oil and water respectively. At the inlet, these three fluids flows in a mixture and at the end of the horizontal pipe, the fluid split into equal proportion to the upward and downward vertical pipe. Based on the water volume fraction, the flow regime at the vertical upward and downward pipe is identified as annular flow regime as water phase occupies the outer layer of wall following by oil and gas in between. This can be supported with analysis of volume fraction at the two vertical outlets which can be found in Appendix B.

Mass flow rates of each fluid at the two outlets are collected, tabulated and compared with the original mass flow rates from Calvalcanti's T-junction design. Differences between the mass flow rates in the two different T-junction designs are later discussed.

Table 13: Mass flow rate for Calvalcanti's and proposed T-junction at two outlets

	Cavalcanti's T-junction		**Proposed T-junction**	
	Outlet 1 (horizontal)	**Outlet 2 (vertical)**	**Outlet downward**	**Outlet upward**
Water	0.1845	0.05110	0.117425	0.118606
Gas	0.0001154	0.0001483	0.000131024	0.000132303
Oil	0.11826	0.10584	0.111429	0.112675

4.1.4 Simulation of proposed t-junction with modified Cavalcanti's fluid model

Modification is implemented to Cavalcanti's fluid model by including gravity effect on the fluid flow in the T-junction. This gravity effect is set to -9.81 m/s acting in the y-direction of the fluid flow. The same simulation parameter was set up and volume fraction of gas, oil and water are produced as shown in Figure 16 below.

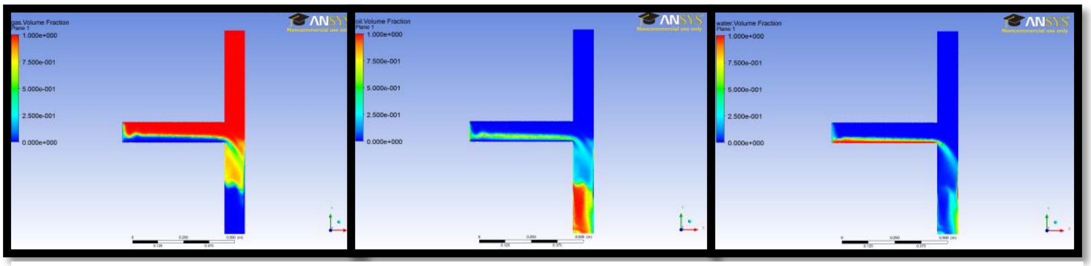


Figure 16: Volume fraction of gas, water and oil in the x-y plane of the proposed T-junction with modified Cavalcanti's fluid model

Figure 16 shows the volume fraction of gas, oil and water respectively. At the inlet, there is a distinctive boundary of gas, oil and water. At the vertical upward outlet, gas is the dominating phase, whereby water and oil dominates the vertical downward outlet. Based on the gas, water and oil volume fraction, the flow regime at the horizontal inlet pipe is identified as stratified flow regime due to the distinctive layer of phases. This can be supported with analysis of volume fraction at the two vertical outlets which can be found in Appendix C. Mass flow rates of each fluid at the two outlets are collected, tabulated and compared with the original mass flow rates from Calvalcanti's T-junction design.

Table 14: Mass flow rate for Calvalcanti's and proposed T-junction at two outlets

	Cavalcanti's T-junction		**Proposed T-junction with modified model**	
	Outlet 1 (horizontal)	**Outlet 2 (vertical)**	**Outlet downward**	**Outlet upward**
Water	0.1845	0.05110	0.305239	0
Gas	0.0001154	0.0001483	8.208 E-0.08	0
Oil	0.11826	0.10584	0.395789	0

4.2 DISCUSSION

4.2.1 Simulation of Azzopardi *et al.* (2000) experiment work

Before moving to the phase split data of the 7 fluid models created, the flow regime was determined. Using the Taitel and Dukler (1976) flow pattern map, the flow regime at the inlet was identified as slug flow. However, Azzopardi (2000) cited that there were no oscillations on the pressure meter or vibrations observed on the pipe. Therefore assumption was made that the flow condition of the inlet fluid will be either stratified or annular regime.

The results presented in Figure 12 for all cases indicate a small level of maldistribution compared to the experiment data. For all cases except case 2 and 4, data shoes that as the flow rates increased, maldistribution decreases. This result is total opposite from other published studies (Rubel *et al.*, 1994 and Azzopardi *et al.*, 1999). This result may due to the homogenising effect set for each case of fluid model as expressed by Azzopardi (2000) which caused the huge error in the value of liquid fraction taken off (L') as shown in Table 9.

4.2.2 Simulation of Cavalcanti *et al.* (2011) simulation study

Despite the fluid model created using the parameters and data provided, there is still error in meshing of geometry in the number of nodes and elements of 0.23% and 2.4% respectively. This error may due the incorrect application of the minimum and maximum mesh face size.

Based on Figure 13, the gas volume fraction dominates at the vertical side-arm, whereas the oil and water remains as mixture and flows to the horizontal outlet, passing the opening of vertical outlet. This is due to the slow flow rate of the fluid. Not surprisingly, the slower flow with larger resident time show better separation (Azzopardi, 1999).

The value of water drop and gas drop diameter was assumed arbitrarily in the computation. The value of 8mm for water drop diameter and 1mm for gas drop diameter is reasonable (Oliveira, 1994).

4.2.3 Simulation of proposed T-junction design with Cavalcanti's fluid model

For this fluid model, the gravity effect was neglected, due to the low flow rates of the fluid flow, which are 0.03m/s for all fluids. By inputting this flow rate into the Taitel and Dukler (1976) flow pattern map, a stratified flow regime is identified. However, Figure 15 indicates otherwise, where the flow regime is a mixture of 3 fluids and there are no distinctive boundaries between the fluids.

Table 13 shows the mass flow rate (kg/s) obtained from the proposed T-junction. It can be noted that there is similar distribution of gas mass flow rate in downward and upward outlets. This is due to the same homogenising effect explained by Azzopardi (2000) due to the low fluid velocity resulting in a laminar flow regime. However, this fluid model is not reasonable as the gas flow rate is much lower than the gas flow rate obtained from the Cavalcanti's T-junction.

4.2.4 Simulation of proposed t-junction with modified Cavalcanti's fluid model

To improve Cavalcanti's fluid model, gravity effect is applied to the fluids in the y-direction (9.81 m/s). This creates a buoyancy effect on the fluid based on the densities of the fluid involved. The result of the gravity effect is shown in Figure 16. At the inlet, there is a distinctive boundary between gas, oil and water phase, creating a stratified flow regime as predicted by Taitel and Dukler (1976) flow pattern map.

Due to the low density and slow velocity of the gas phase, gas phase tend to flow upwards into the vertical pipe as predicted by Kolnes and Ashiem (1990) and Katsaounis *et al*. (1997), who did a study of phase split with a vertical upward branch arm in the main T-junction pipeline. The heavier phase, which the oil and water phase tend to travel downward to the vertical outlet due to its higher density compared to gas. If the velocity is to be increased of all the fluid, theory prediction by Shoham *et al*, (1997) suggest that there will be water and oil intake in the vertical upward outlet and the rate of intake increases as more gas been taken-off into the vertical upward outlet.

Despite the impressive 100% domination of the gas phase in the vertical upwards outlet, this modified fluid model is not reasonable and cannot be applied, since there is now mass flow rate recorded in the upward outlet as shown in Table 13. This may due to the setting of the CDF model domain, which suggest that the outlets are set as opening, which allow two way flows (including reversible flow). Further alterations can be made to improve the model and is stated in more detail in the recommendation section.

CHAPTER 5
CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

Based on the study conducted, the experiment fluid model is not applicable for the study of the proposed T-junction design. This is due to the high percentage error obtained and many assumptions were made, resulting in a development of unstable fluid model. For the simulation work fluid model, the results obtained from it are reasonable and applicable for the phase split study on the proposed T-junction design despite very small percentage error in the meshing section. After the application of the Cavalcanti's fluid model into the proposed T-junction design, the volume fraction of gas, water and oil are not encouraging.

Thus modification is applied to the model by adding the gravity effect of on the fluids involved creating a buoyancy effect and as expected there are significant improvements in the phase volume fraction. The gas volume fraction obtained from the proposed T-junction indicates a 100% gas intake into the vertical upward outlet. However, the gas mass flow rate was not detected in any of the outlets, indicating vacuum region in the gas volume fraction zone.

The new proposed T-junction is a novel design and has not been attempted by any researchers till date. More studies and simulation need to be performed to better analyze and understand the phase split phenomena at the new T-junction design. The current fluid model with the gravity effect applicable but more alterations need to be implemented to obtain a 100% gas mass flow rate result. Future recommendation, will allow better development of fluid model and better understanding of the efficiency of the new proposed T-junction design.

5.2 RECOMMENDATION

There are few recommendation suggested that can be implemented in the future for a better computation fluid dynamics study on the two-phase separation in a T-junction.

1. Cavalcanti's fluid model has been modified with application of gravity effect on each fluid phase. The further modification suggested is increasing the superficial velocity of the each fluid, which allows the application of turbulent flow model and accounts for the high phase maldistribution at the impact at the end of the horizontal inlet pipe.

2. The current fluid model can be improved by reducing the outlets pressure to 1 atm and set the reference pressure at 0 atm, to allow the simulation to calculate the pressure difference using the volume fraction and velocity of the each fluid. The higher the pressure drop, the more the fraction of liquid/gas is directed/taken-off to the vertical outlets.

3. The current proposed T-junction design is a regular T-junction with similar diameter openings for inlet and 2 outlets. A better separation might possible with a reduce outlet diameter opening. Two geometrical designs are suggested as shown in Figure 17 below.

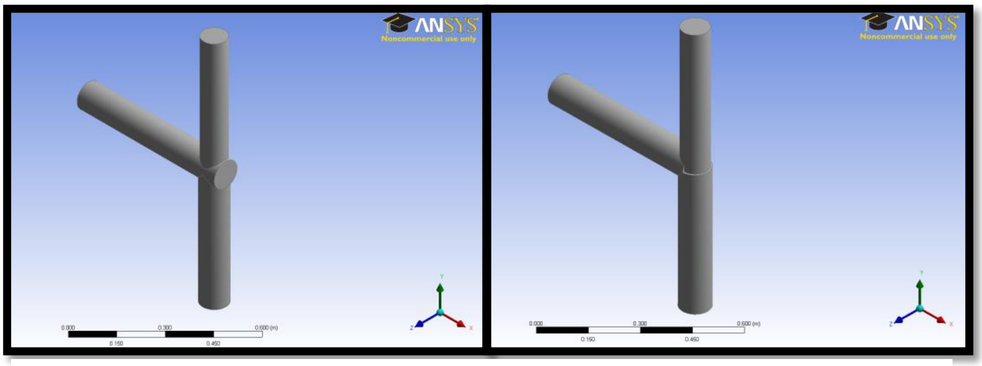


Figure 17: Proposed reduced T-junction design

REFERENCE

1. Alboudwarej, H., Huo, Z., and Kempton, E., (2006) "*Flow-Assurance Aspects of Subsea Systems Design for Production of Waxy Crude Oils*", SPE Annual Technical Conference and Exhibition, San Antonio, Texas, U.S.A., 24–27 September 2006.

2. Adechy, D. (2000), PhD Thesis, Imperial College of Science, Technology and Medicine, London.

3. Adechy, D. and Issa, R. I. (1999), "*Numerical modelling of horizontal annular flows*", 2nd edition Int. Symp on Two-phase Flow Modelling and Experimentation, Pisa (Italy), 23-26 May, 1607- 1615.

4. Amir, T.A.: "The spilt of horizontal two-phase flow at a T-junction – CFD Study," Thesis, University of Malaya.

5. Azzopardi, B. J. (1999a), "The effect of side arm diameter on phase split at T-junctions", *SPE Annual Technical Conference and Exhibition*, Houston, Texas.

6. Azzopardi, B. J. (1999b), "Phase separation at T-junctions", *Multiphase Science and Technology*, Vol. 11, pp. 223-329.

7. Azzopardi, B. J. and Hervieu, E. (1994), "Phase separation at T-junctions", *Multiphase Science and Technology*, Vol. 8, pp. 645-714.

8. Azzopardi, B. J. (1984), The effect of side arm diameter on two-phase split at a T-junction, *International Journal of Multiphase Flow*, 10, 509-512.

9. Azzopardi, B. J., Colman, D. A., and Nicholson, D. (2002), "*Plant application of a T-junction as a partial phase separator*", Transactions of the Institute of Chemical Engineers, Vol. 80A, pp. 87-96.

10. Azzopardi, B. J., Rea S. (2000), "*Phase Separation Using a Simple T-junction*", SPE 63040, University of Nottingham, Nottingham, England.

11. Azzopardi, B. J., Whalley, P. B. (1982), "*The effect of flow pattern on two-phase flow in a T-junction*", International Journal of Multiphase Flow, 8, 491-507.

12. Baker, G. (2003), "Separation and control of gas-liquid flows at horizontal T-junctions", PhD Thesis, University of Nottingham, Nottingham, England.

13. Baker, O. (1954), "*Simultaneous flow of oil and gas*", Oil and Gas Journal, Vol. 53, pp. 385-389.

14. Ballyk, J. D., Shoulry, M. And Chan, A. M. C. (1988), "*Steam-water annular flow in a horizontal dividing T-junction*", International Journal of Multiphase Flow, 14, 265-285.

15. Barnea, D. and Taitel, Y. (1985), "Flow pattern transition in two-phase gas-liquid flows", Encyclopaedia of Fluid Mechanics, Vol. 2, Chapter 16, Gulf Publishing Company, Houston, Texas.

16. Beggs, H.D. and Brill, J.P., 1973: "A Study of Two-phase Flow in Inclined Pipe," Journal of Petroleum Technology, May.

17. Cavalcanti, W. S., Belem, E. Z. G., de Lima, W. C. P. B., Cavalcante, F. P., Barbosa, E. S., Farias Neto, S. R. (2011), "*Non-isothermal 3 phase flow of Petroleum, Gas, and Water in T and Y junctions*", International Journal of Modelling and Simulation for Petroleum Industry, Vol. 5, No. 1, pp. 43-52.

18. Charron, Y., and Whalley, P. B. (1995), "*Gas-Liquid annular flow at a vertical T-junction-part1: Flow Separation*", International Journal of Multiphase Flow, 21, 569-589.

19. Chisholm, D. (1972): “An equation for velocity ration in two-phase flow”, NEL Report No. 533.

20. Christopher, E. B. (2005), “*Fundamentals of Multiphase Flows*”, PhD Thesis, California Institute of Technology.

21. Conte, G. (2000), “*An Experiment Study for the Characterisation of Gas/Liquid Flow Splitting at T-junctions*”, PhD Thesis, University of Nottingham, Nottingham, England.

22. Hong, K. C. (1978), “Two-phase flow splitting at a pipe tee ”, Journal of Petroleum Technology, Vol. 2, pp. 290-296.

23. Hwang, S. T., Soliman, H. M., and Lahey, R. T. (1988), “Phase separation in dividing two-phase flows, International Journal of Multiphase Flow, 14, 439-458.”

24. Moreno J. Q. (2005), “Experimental and Analytical study of two-phase pressure drops during evaporation in horizontal tubes”, PhD Thesis, Universite Pierre et Marie Curie, France.

25. Lahey, R. T. (1986), “*Current understanding of phase separation mechanism in branching conduits*”, Nuclear Engineering and Design, Vol 95, pp. 145-161.

26. Mandhane, J. M., Gregory, G. A., and Aziz, K. (1974), “A flow pattern map for gas-liquid flow in horizontal pipes”, International Journal of Multiphase Flow, Vol. 1, pp. 537-553.

27. Mak, C. Y., Omebere-Iyari, N. K. And Azzopardi B. J. (2006), “The split of vertical two-phase flow at a small diameter T-junction”, Chemical Engineering Science, Vol. 61, No. 19, pp. 725-748.

28. Mohammed, A., Prof. Niranjan, K. I., Ass. Prof. Sadoun, F.D. (2012), "*CFD simulatios and flow analysis through a T-junction pipe*", International Journal of Engineering Science and Technology (IJEST), Vol. 4, No.7, pp. 3392-3407.

29. Oranje, L (1973), Condensate behaviour in gas pipeline is predictable, *Oil and Gas* J., 71, 39-44.

30. Ottens, M., Hoefsloot, H. C. J., and Hamersa, P. J. (1999), "Effect of small branch inclination on gas-liquid flow separation in T-junctions", AlChe Journal, Vol. 45, No. 3, pp. 465-474.

31. Peng, F., Shoukri, M., and Ballyk, J. D. (1998), "*An experiment investigation of startified steam-water flow in a T-junction*", 3rd International Conference on Multiphase Flow, Lyon, France.

32. Rea, S. and Azzopardi, B.J.: "The split of horizontal stratified flow at a large diameter T-junction", submitted to Trans ASME, J.Fluids Eng. (2000).

33. Rea, S. (1998), "*Stratified flow at T-junctions*", PhD Thesis, Nottingham University.

34. Reimann, J., Brinkmann, H. J., and Domansky, R. (1988), "*Gas-Liquid flow in dividing T-junctions with horizontal inlet and different branch orientations and diameters*", Kernforschungszentrum Karlsruhr, Institute fur Reactorbauelemente, Report KfK 4399.

35. Robert, PA (1994), *Two-phase flow at T-junctions*", PhD Thesis, Nottingham University.

36. Shelley, S., "Computational Fluid Dynamics – Power to the Peeple," Chem. Eng. Prog. 103(4), 10-13 (2007).

37. Shoham, O., Brills, J. P., and Taitel, Y. (1987), "*Two-phase flow split in a T-junction – Experiment and Modelling*", International Journal of Multiphase Flow, 42,2667-2676.

38. Shoham, O., Arirachakaran, S., Brill, J. P. (1989), "Two-phase flow splitting in a horizontal reduced pipe tee", Chemical Engineering Science, Vol. 10, pp. 2388-2391.

39. S.Z.Rouhani and M.S. Sohel, "Two-phase Flow Pattern: A Review of Research Result," *Progress in Nuclear Energy,* pp 217-259,1983

40. Taite, Y. and Dukler, A.E.: "A model for predicting flow regime transitions in horizontal and near-horizontal gas-liquid flow", *AIChEJ*, (1976) 22, 47.

41. Walter, L. C., Soliman, H. M., and Sims, G. E. (1998), "*Two-phase pressure and phase distribution at reduced T-junctions*", International Journal of Multiphase Flow, 24, 775-792.

42. Wren, E. (2011), "Geometry effect on phase split at a large diameter T-junction", PhD Thesis, University of Nottingham, Nottingham, England.

43. Yang, L., Zhao, Z., Qi, P., Zhao, L., & Azzopardi, B. J. (2010). Phase Separation of Gas-Liquid Two Phase Flows in Multi-tube T-junction Separator. *The Second China Energy Scientist Forum* , 241-244.

APPENDIX A: FLUID MODEL

EXPERIMENT FLUID MODEL

1. BASE MODEL

	A	B
1		**Case 1 - Base**
3	Basic Setting	Fluid: Gas & Liquid
4		Non-bouyant
5		
6		Domain Reference Pressure: 1 atm
8	Fluid Model	Turbulance model: Homogenous
9		k-epsilon model
10		
11		
13	Fluid Pair Model	Interface transfer: mixture model
14		Surface tension coefficient: 0.015 (N/m)
15		Drag coefficient: 0.44
16		
18	Fluid properties	Bulk mass flowrate: 2.7296 (kg/s)
19		velocity water = 0.74 (m/s)
20		velocity Air = 0.76 (m/s)
21		density water: 790 (kg/m3)
22		density Air = 22.5 (kg/m3)
23		volume fraction: Air = 0.51
24		volume fraction Water = 0.49
26	Outlet Input	Horizontal outlet relative pressure: 1 atm
27		Vertical outlet relative pressure: 1 atm
29	Assumptions	No slip wall
30		Smooth Wall
32	Solver Setting	Max Interation: 300
33		Timescale: 1
34		Residual Target: 1E-4

2. BOUYANT_MAXIMUM DENSITY MODEL

	A	D
1		**Case 3: Bouyant_max density**
3	Basic Setting	Fluid: Gas & Liquid
4		Bouyant: Bouyant reference:- 790 (kg/m
5		Bouyancy Model: Density difference
6		Domain Reference Pressure: 1 atm
8	Fluid Model	Turbulance model: Homogenous
9		k-epsilon model
10		
11		
13	Fluid Pair Model	Interface transfer: mixture model
14		Surface tension coefficient: 0.015 (N/m)
15		Drag coefficient: 0.44
16		
18	Fluid properties	Bulk mass flowrate: 2.7296 (kg/s)
19		velocity water = 0.74 (m/s)
20		velocity Air = 0.76 (m/s)
21		density water: 790 (kg/m3)
22		density Air = 22.5 (kg/m3)
23		volume fraction: Air = 0.51
24		volume fraction Water = 0.49
26	Outlet Input	Horizontal outlet relative pressure: 1 atr
27		Vertical outlet relative pressure: 1 atm
29	Assumptions	No slip wall
30		Smooth Wall
32	Solver Setting	Max Interation: 300
33		Timescale: 1
34		Residual Target: 1E-4

3. BOUYANT_MINIMUM DENSITY MODEL

1		**Case 2: Bouyant_min density**
3	Basic Setting	Fluid: Gas & Liquid
4		Bouyant: Bouyant reference:- 22.5 (kg/m3
5		Bouyancy Model: Density difference
6		Domain Reference Pressure: 1 atm
8	Fluid Model	Turbulance model: Homogenous
9		k-epsilon model
10		
11		
13	Fluid Pair Model	Interface transfer: mixture model
14		Surface tension coefficient: 0.015 (N/m)
15		Drag coefficient: 0.44
16		
18	Fluid properties	Bulk mass flowrate: 2.7296 (kg/s)
19		velocity water = 0.74 (m/s)
20		velocity Air = 0.76 (m/s)
21		density water: 790 (kg/m3)
22		density Air = 22.5 (kg/m3)
23		volume fraction: Air = 0.51
24		volume fraction Water = 0.49
26	Outlet Input	Horizontal outlet relative pressure: 1 atm
27		Vertical outlet relative pressure: 1 atm
29	Assumptions	No slip wall
30		Smooth Wall
32	Solver Setting	Max Interation: 300
33		Timescale: 1
34		Residual Target: 1E-4

4. DISPERSED GAS MODEL

1		**Case 4: Dispersed Gas**
3	Basic Setting	Fluid: Gas & Liquid
4		Non-bouyant
5		
6		Domain Reference Pressure: 1 atm
8	Fluid Model	Turbulance model: Homogenous
9		k-epsilon model
10		Gas as dispered flow
11		Liquid as continous flow
13	Fluid Pair Model	Interface transfer: Dispersed model
14		Surface tension coefficient: 0.015 (N/m)
15		Drag coefficient: 0.44
16		Interphase transfer: Free surface
18	Fluid properties	Bulk mass flowrate: 2.7296 (kg/s)
19		velocity water = 0.74 (m/s)
20		velocity Air = 0.76 (m/s)
21		density water: 790 (kg/m3)
22		density Air = 2.25 (kg/m3)
23		volume fraction: Air = 0.51
24		volume fraction Water = 0.49
26	Outlet Input	Horizontal outlet relative pressure: 1 atm
27		Vertical outlet relative pressure: 1 atm
29	Assumptions	No slip wall
30		Smooth Wall
32	Solver Setting	Max Interation: 300
33		Timescale: 1
34		Residual Target: 1E-4

5. FREE SURFACE MODEL

	A	E
1		**Case 5: Free Surface Model**
3	Basic Setting	Fluid: Gas & Liquid
4		Non-bouyant
5		
6		Domain Reference Pressure: 1 atm
8	Fluid Model	Turbulance model: Homogenous
9		k-epsilon model
10		Free Surface Model
11		Interphase Compression: 2
13	Fluid Pair Model	Interface transfer: mixture model
14		Surface tension coefficient: 0.015 (N/m)
15		Drag coefficient: 0.44
16		Interphase transfer: Free surface
18	Fluid properties	Bulk mass flowrate: 2.7296 (kg/s)
19		velocity water = 0.74 (m/s)
20		velocity Air = 0.76 (m/s)
21		density water: 790 (kg/m3)
22		density Air = 2.25 (kg/m3)
23		volume fraction: Air = 0.51
24		volume fraction Water = 0.49
26	Outlet Input	Horizontal outlet relative pressure: 1 atr
27		Vertical outlet relative pressure: 1 atm
29	Assumptions	No slip wall
30		Smooth Wall
32	Solver Setting	Max Interation: 300
33		Timescale: 1
34		Residual Target: 1E-4

6. FREE SURFACE AND BOUYANT_MAXIMUM DENSITY MODEL

	A	E
1		**Case 6: Free Surface Model + Bouyant_max**
3	Basic Setting	Fluid: Gas & Liquid
4		Bouyant: Bouyant reference:- 790 (kg/m3)
5		Bouyancy Model: Density difference
6		Domain Reference Pressure: 1 atm
8	Fluid Model	Turbulance model: Homogenous
9		k-epsilon model
10		Free Surface Model
11		Interphase Compression: 2
13	Fluid Pair Model	Interface transfer: mixture model
14		Surface tension coefficient: 0.015 (N/m)
15		Drag coefficient: 0.44
16		Interphase transfer: Free surface
18	Fluid properties	Bulk mass flowrate: 2.7296 (kg/s)
19		velocity water = 0.74 (m/s)
20		velocity Air = 0.76 (m/s)
21		density water: 790 (kg/m3)
22		density Air = 2.25 (kg/m3)
23		volume fraction: Air = 0.51
24		volume fraction Water = 0.49
26	Outlet Input	Horizontal outlet relative pressure: 1 atm
27		Vertical outlet relative pressure: 1 atm
29	Assumptions	No slip wall
30		Smooth Wall
32	Solver Setting	Max Interation: 300
33		Timescale: 1
34		Residual Target: 1E-4

7. FREE SURFACE AND BOUYANT_MINIMUM DENSITY MODEL

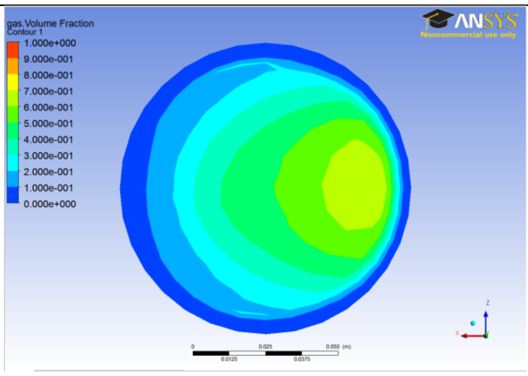
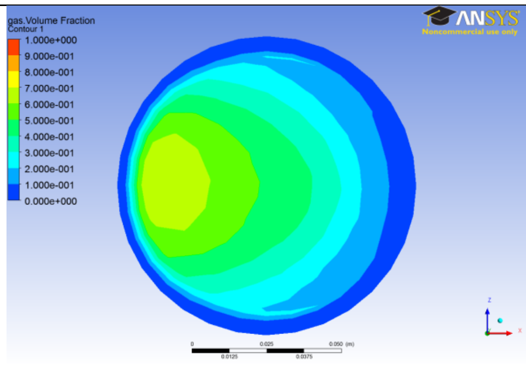
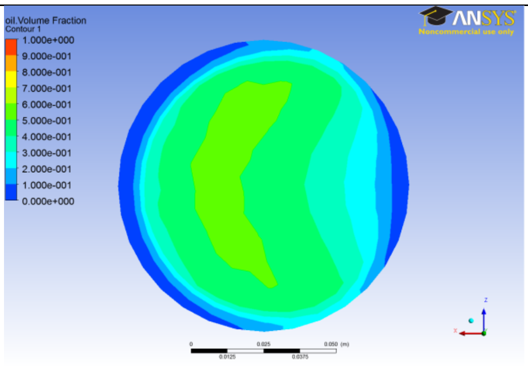
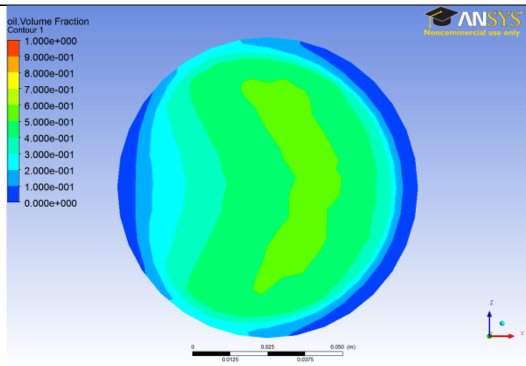
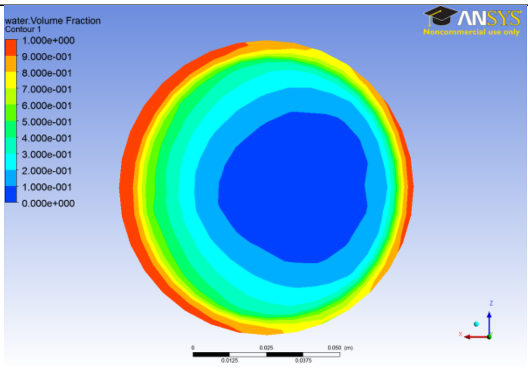
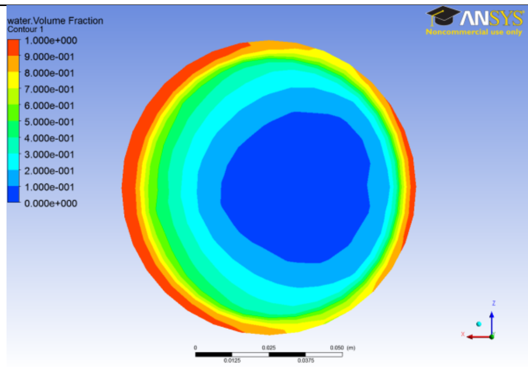
	A	E
1		**Case 6: Free Surface Model + Bouyant_max**
3	Basic Setting	Fluid: Gas & Liquid
4		Bouyant: Bouyant reference:- 22.5(kg/m3)
5		Bouyancy Model: Density difference
6		Domain Reference Pressure: 1 atm
8	Fluid Model	Turbulance model: Homogenous
9		k-epsilon model
10		Free Surface Model
11		Interphase Compression: 2
13	Fluid Pair Model	Interface transfer: mixture model
14		Surface tension coefficient: 0.015 (N/m)
15		Drag coefficient: 0.44
16		Interphase transfer: Free surface
18	Fluid properties	Bulk mass flowrate: 2.7296 (kg/s)
19		velocity water = 0.74 (m/s)
20		velocity Air = 0.76 (m/s)
21		density water: 790 (kg/m3)
22		density Air = 2.25 (kg/m3)
23		volume fraction: Air = 0.51
24		volume fraction Water = 0.49
26	Outlet Input	Horizontal outlet relative pressure: 1 atm
27		Vertical outlet relative pressure: 1 atm
29	Assumptions	No slip wall
30		Smooth Wall
32	Solver Setting	Max Interation: 300
33		Timescale: 1
34		Residual Target: 1E-4

SIMULATION FLUID MODEL

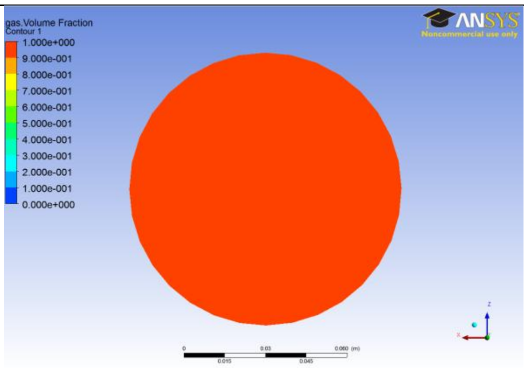
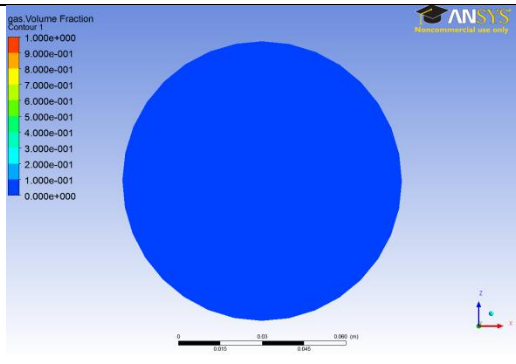
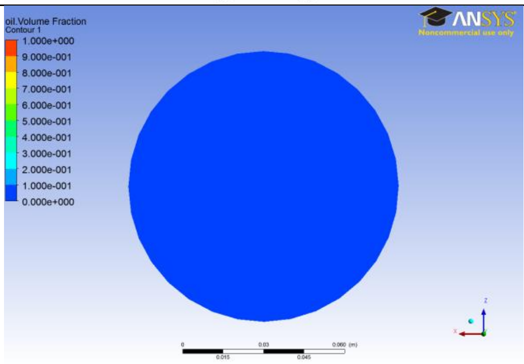
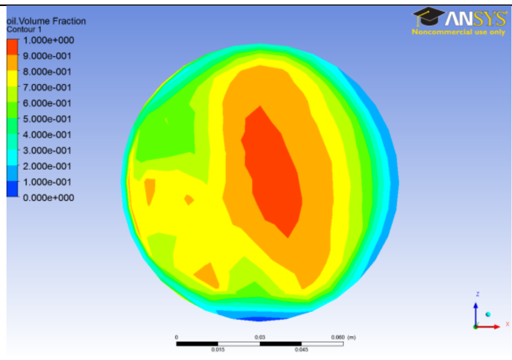
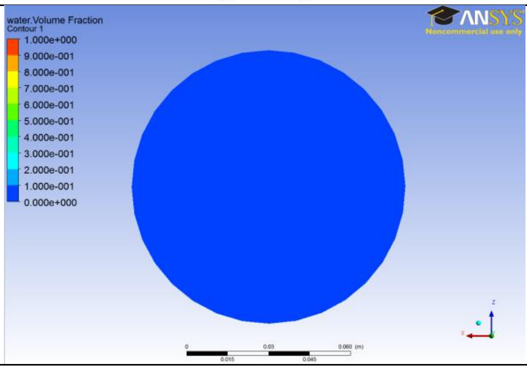
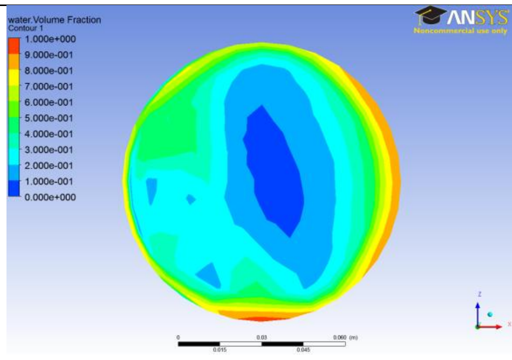
1. BASE MODEL

	A	F
1		**Case 1: Base**
3	Basic Setting	Fluid: Gas, Oil & Water
4		Non-bouyant
5		
6		Domain Reference Pressure: 1 atm
8	Fluid Model	Non-homogenous
9		Mean diamter of gas 1mm and water 8mm
10	**1.**	Gas as dispered flow
11		Oil and Water as continous flow
13	Fluid Pair Model	Interface transfer: Dispersed model
14		Surface tension coefficient: 0.015 (N/m)
15		Drag coefficient: 0.44
16		Laminar Flow
18	Fluid properties	Velocity of gas-oil-water = 0.03 (m/s)
19		density of gas = 1.12 (kg/m3)
20		density of oil = 951 (kg/m3)
21		density water: 1000 (kg/m3)
22		volume fraction of gas = 0.1
23		volume fraction od oil = 0.7
24		volume fraction of Water = 0.2
26	Outlet Input	Horizontal outlet relative pressure: 101.326 kPa
27		Vertical outlet relative pressure: 101.326 kPa
29	Assumptions	No slip wall
30		Smooth Wall
32	Solver Setting	Max Interation: 1000
33		Timescale: 1
34		Residual Target: 1E-7

APPENDIX B: VOLUME FRACTION OF OUTLETS FOR PROPOSED T-JUNCTION WITH CAVALCANTI'S MODEL

<table>
<tr><th>Vertical Upward Outlet</th><th>Vertical Downward Outlet</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

APPENDIX C: VOLUME FRACTION OF OUTLETS FOR PROPOSED T-JUNCTION WITH MODIFIED CAVALCANTI’S MODEL

<table>
<tr><th>Vertical Upward Outlet</th><th>Vertical Downward Outlet</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>