CERTIFICATION OF APPROVAL

Effects of Void Fractions on Vibration Behaviour of Pipe Conveying Fluids

by

MUHAMMAD SYAMIM BIN MOHD SAIFUDDIN

24339

A project dissertation submitted to

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

In partial fulfilment of the requirement for the

BACHELOR OF MECHANICAL ENGINEERING

WITH HONOURS

Approved by,

(Dr. Nabihah Sallih)

UNIVERSITI TEKNOLOGI PETRONAS

SERI ISKANDAR, PERAK

JANUARY 2021

CERTIFICATION OF ORIGINALITY

This is to certify that I am solely accountable for the research analysis submitted throughout this project, that the original work is my own (except as noted in the references and acknowledgements), and that the original work found herein has not been carried out or performed by any other sources or persons.

____

MUHAMMAD SYAMIM SAIFUDDIN

ABSTRACT

Topology in a piping system is an important aspect to be considered for designing a safety piping system. Without proper consideration towards piping topologies, Vibration from the flow itself can be generated and causes failure towards the system. This occurrence is called Flow Induced Vibration (FIV). It is a common event that happens in many industries and the problem has not been given much attention until today. Nowadays people are looking for ways to prevent it from happening as it clearly shows it effectiveness in damaging the piping system. During the design process of a plant, FIV is difficult to predict, and it is typically first observed during the activity phase. This study investigates the effect of piping topologies of two-phase flow on the vibration response in a piping system using simulation method.

Computational Fluid Dynamics (CFD) analysis was performed to compute the results for this research study. Before anything, validation based on previous paper is needed to ensure the results is reliable and can be used to produce legitimate results. Flow behaviour of a vertical straight pipe with an upward flow was obtained for validation from using Altair AcuSolve and Altair Hyperworks. In this validation, 2 different level set settings which were conventional level set and Back and Forth Error Compensation and Correction (BFECC) have been compared to the published results from the past paper. It is observed that BFECC method was quite similar to the desired result as in the past paper. However, due to time given to complete this research project, it was not possible to investigate the vibration response of the pipe yet.

ACKNOWLEDGEMENT

First and Foremost, I would like to express my gratitude to the almighty for blessing me with strength and health throughout my entire final year project journey. Next, I am grateful to have Dr Nabihah Sallih as my supervisor. She was a great mentor to me and constantly providing me knowledge and guidance throughout my journey. Even with rough schedule, she always finds time for her students. Moreover, I would like to also express my gratitude towards Universiti Teknologi PETRONAS for providing me all the items I need in completing the project such as access to the remote PC. Furthermore, I would like to express my great appreciation to the coordinator, Dr Jundika for all the knowledge and guidance that given to me during the seminars. I am able to complete my project with the right technique and tips. Last but not least, thanks to my family and friends who always with me through ups and downs throughout the process.

TABLE OF CONTENTS

CERTIFICATION OF APPROVAL i

CERTIFICATION OF ORIGINALITY ii

ABSTRACT iii

ACKNOWLEDGEMENT iv

TABLE OF CONTENTS v

LIST OF FIGURES vii

LIST OF TABLES viii

CHAPTER 1 9

INTRODUCTION 9

1.1 Background 9

1.2 Problem Statement 10

1.3 Objective 11

1.4 Scope of Study 11

CHAPTER 2 12

LITERATURE REVIEW 12

2.1 Flow Patterns 13

2.2 Effect of void fraction & Volumetric Quality 15

2.3 Effect of Pipe Geometry 16

CHAPTER 3 18

METHODOLOGY 18

3.1 Flowchart Project 18

3.2 Reference Validation Set up 20

3.3 Model Geometry for Validation 22

3.4 Boundary Condition for Validation 23

3.5 Meshing for Validation 24

3.5 Materials Piping for Validation 25

3.6 Level Set Approach 25

3.7 Key Milestones 27

3.7.1 Final year Project 1 (FYP 1) 27

3.7.2 Final year Project II (FYP I1) 28

CHAPTER 4 31

RESULT AND DISCUSSION 31

4.1 Convergence Study 31

4.2 Flow Regime Identification 33

4.3 Time evolution of the flow 34

4.4 Comparison Literature and Validation Model with Both Approaches 35

CHAPTER 5 39

CONCLUSIONS AND RECOMMENDATIONS 39

5.1 Conclusion 39

5.2 Recommendations 40

LIST OF FIGURES

Figure 1.1: Piping System Leakage 9
Figure 2.1: Flow regimes for two-phase flow in a vertical pipe with upward flow direction. Adapted from [10]........ 14
Figure 2.2: Flow regimes for two-phase flow in a horizontal pipe. adapted from [10] 15
Figure 3.1: Project Flow Chart........ 19
Figure 3.2: Validation Geometry from Literature [10] 21
Figure 3.3: Validation Mesh from Literature [10] 21
Figure 3.4: Model Geometry for Validation 23
Figure 3.5: Perturbation Set up 23
Figure 3.6: Structured Mesh........ 25
Figure 3.7: FYP I Milestone........ 27
Figure 3.8: FYP II Milestone 28
Figure 4.1: Residual Ratio for Standard Level set approach 32
Figure 4.2: Residual Ratio for BFECC approach 32
Figure 4.3: Baker's Chart for Vertical Pipe [10] 33

LIST OF TABLES

Table 3.1: Under Relaxation Factors [10] 22
Table 3.2: Density of Air, water, and Perspex 24
Table 3.3: Gantt Chart for FYP 1 29
Table 3.4: Gantt Chart for FYP 2 30
Table 4.1: Time Evolution of the air and water contour in the vertical pipe based on air volume fraction. 34
Table 4.2: Comparison 1 of experimental data VS Conventional and BFECC Level set 36
Table 4.3: Sample of Comparison of Cross Section between Published Experiment VS Published Simulation VS Conventional Level set Validation Model VS BFECC Validation Model 38

CHAPTER 1

INTRODUCTION

1.1 Background

In this modern era, piping system plays an important role in many industries not just limited to oil and gas industries, but also other industries such as nuclear plant, transportation and many more. However, accidents and leakages in piping industry involving the multiphase flow have increased massively throughout the year as shown in figure 1.1. In the chemical, petroleum, manufacturing, nuclear and power industries, the piping system that conveys two-phase flow involving gas and liquid has been widely applied. As for nuclear and power industries, two-phase flows are generally used for two-phase boiling, whereas those carried out in the oil and chemical industries are essentially non-boiling in nature. Nevertheless, because of its failures and accidents, the main characters of the industries such as offshore oil production found tremendous losses in terms of human life and industry.

FIGURE 1.1: Piping System Leakage

Flow induced Vibration (FIV) can be the main factor that causes the failure in the piping systems that convey two-phase flow such as gas and liquid. In engineering implementation, FIV becomes more critical due to internal flowing fluid awareness with fluctuating powers. FIV can cause many forms of failure today and become disruptive to the operation. Multiphase flows are difficult to study, since its behaviours are unsteady and unstable. Therefore, FIV prediction studies and their interaction with the piping structure are particularly crucial in the oil and gas industries.

Generally, the study of vibration analysis under different topologies of piping system is not widely researched yet. Therefore, given the circumstances. This project has been given the opportunity to investigate the vibration response under different piping topologies conveying two-phase flow.

1.2 Problem Statement

Internal structural flow induced vibration has been found in a number of sectors, including heat exchange tubing, chemical plants, nuclear reactors, and subsea pipeline construction. Many Cases involving flow induced pipe vibration have been observed over the years. It has become one of the common problems in the piping system of plants. Several experiments have been performed to explain the vibration caused by flow, but the question of how the vibration functions still needs to be thoroughly understood. In the early stage of design, due to the complication of flow induced vibration, it is to be expected and only noticeable after the operation has begun. The plant’s high output and less shutdown put a lot of pressure on the piping. The issue become worsen as maintenance has been neglected causing accidents when failure during operation.

Although several studies have been performed, there is still limited understanding of two-phase flow in a wide variation of piping topology. There is still no detailed overall model of comparison between topologies conveying two-phase flow that can be used in the analysis and design of a safety piping system due to the unstable aspect of the two-phase flow. The studies that were carried out focused only on another factor, such as fluid or force characteristics, but there is still less research that focuses on vibration response.

1.3 Objective

The aim of this is to investigate the effects of piping topologies and its corresponding direction on vibration response using simulation method and to compare vibration amplitude and its corresponding frequency obtained from simulation result with the experimental result.

1.4 Scope of Study

This paper stresses the study of the influence of various topologies on the frequency and amplitude of the two-phase flow conveying pipe vibration. This paper is carried out using a system of simulation, such as ANSYS. For verification purposes, this analysis will include experimental findings from previous internal research as a result of comparison. The software involve in this study is ANSYS with three different analysis which are CFD Fluent, Transient structural analysis and system coupling. For CFD fluent, fluid behaviour will be analysed. Whereas transient structural will produce natural frequency and vibration amplitude as results. System coupling will be used to couple result from CFD into transient structural.

CHAPTER 2

LITERATURE REVIEW

Theory implements in this research study to achieve the correct outcome was discussed in this section. Moreover, study on previous research by many authors was conducted to improve the understanding of the topic of effect of topologies on vibration response. Many studies have been done experimentally as well as numerically. According to Miwa et al. [1], two-phase internal flow studies are needed to achieve the maximum level of piping system efficiency and integrity. However, flow induced vibration (FIV) has been the topic of just a few studies and analysis. The number of studies comparing different topologies is especially small. The flow turning components in the piping system are one of the key causes of change in momentum flux in FIV. It has been found that vibration tends to occur during slug flow regime compared to other regimes.

Fujikawa et al. [2] stated that vibration amplitudes during slug flow regime are high. It is, however, less reliant on the multiphase flow velocity. The vibration amplitudes of the plug flow regime, on the other hand, are influenced by the flow rate, considering its limited value. On the other aspect of pipe design, annular flow is considered to be the best flow regime as it has stable pattern that led to less or no piping vibration. However, vertical pipe conveying fluids can be very unstable due to the pressure drops as the liquid moves upward

2.1 Flow Patterns

Flow patterns or flow regimes is also one of the several factors to be considered in a pipe conveying fluids. Different variation of flow patterns will result in different behaviour or characteristics of the fluid flow. Different flow regimes will occur with different gas and liquid flow rates as well as different topologies and its flow directions. Therefore, it is crucial to know the flow rates in order to predict the flow regimes as part of the safety measures for the flow piping systems.

As this research focuses on vertical straight pipe with upward flow direction, the flow regimes of the stated case are shown in figure 2.1. The sequence is as gas flow rates increases. According to Holland et al. [12], bubbly flow will occur at lower gas flow rate and as the gas flow rate increases, the average bubble size gets larger. As a result, following flow regime occurs when the gas flow speeds rise to the point that several bubbles join together to form slugs of gas. Furthermore, this statement is further supported by an article stated by Agrawal et al. [10], in which it is stated that bubbly and slug flow are form at lower and intermediate gas flow rates, respectively. However, these patterns will be destroyed at higher flow rates and a chaotic type of flow forms generally knows as churn flow. In this phase of flow pattern, there is a churning motion of irregularly arranged gas liquids sections around the majority of the cross section. Moreover, by further increasing its gas flow rate, the phases begin to separate, with the liquid running all around the pipe’s wall and the gas in the centre.

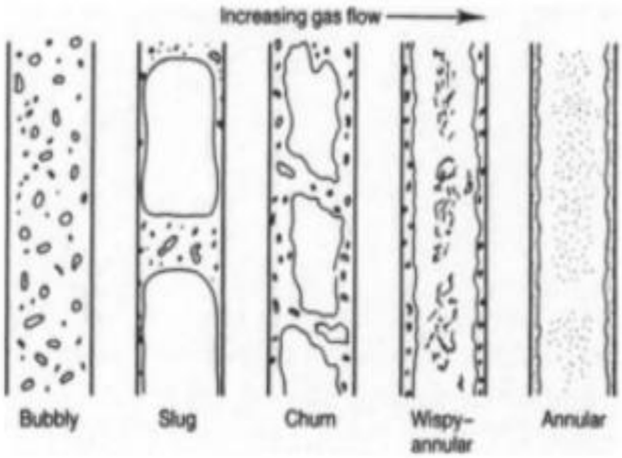


FIGURE 2.1: Flow regimes for two-phase flow in a vertical pipe with upward flow direction. Adapted from [10]

In a different case of topologies such as horizontal pipe, Holland et al. [12] stated that the patterns of gas-liquid flow in horizontal pipes are identical to those seen in vertical pipes. Asymmetry, on the other hand, is induced by gravity which is more noticeable at low flow rates. The order of flow regimes is shown in figure 2.2. According to the figure, the bubbles in the bubbly regime are limited to an area near the top of the pipe. As the gas flow rate is increased, the bubbles become larger and join to create long bubbles, resulting in the formation of plug flow regime. The gas plugs join to form a single gas later in the upper part of the pipe at higher gas flow rates. Which then result in the smooth interface between gas and liquid. This type of occurrence is called stratified flow. The formation of wavy flow is then generated when gas flow rate is further increased to the point that the interfacial shear stress is strong enough to produce waves on the liquid's surface. As when the gas flow rate continues to increase, slug regime is formed when the waves that move in the direction of the flow intensify in size until their crests reach the pipe's wall, where the gas bursts through and liquid is distributed across the pipe's wall. Up to this point, the regime will continue to transition into another regime with increasing gas flow rates according to the figure. An annular regime similar to vertical pipe with upward flow is then observed at a higher gas flow rate in which liquid flow at the wall of the pipe and gas flows through the centre of the pipe. However, if the flow rates are increased further, very thin liquid film will be produced, and this flow is called spray or mist flow regime.

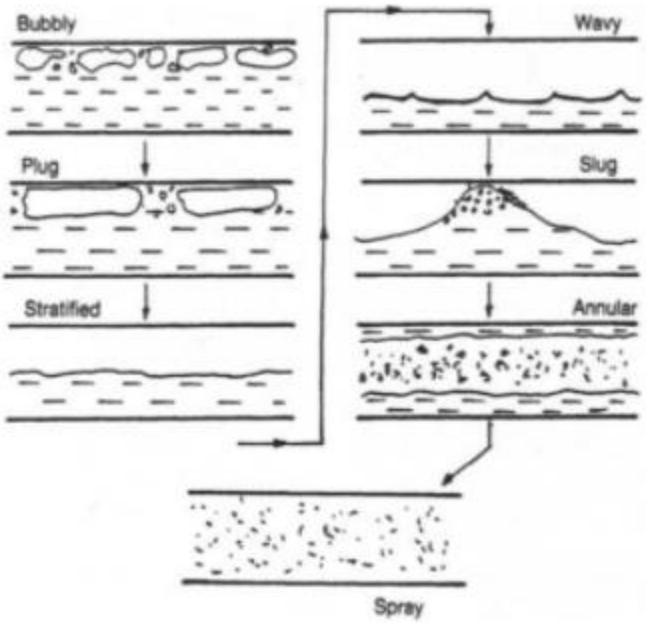


FIGURE 2.2: Flow regimes for two-phase flow in a horizontal pipe. adapted from [10]

2.2 Effect of void fraction & Volumetric Quality

At high volumetric quality levels, the power density spectrum exhibited strong peaks, while at lower volumetric quality ranges, such as $\beta \leq 70$, it produced differing peaks. Due to slug, annular flow and slug flow in particular surface waves with amplitude and significant momentum variations were generated in the high volumetric quality range. At lower range of volumetric quality, these parameters' magnitudes were not constructive. [3]. Volumetric quality is described as below.

$$\beta = \frac{Q_g}{Q_g + Q_f} \tag{2.1}$$

$\beta = Volumetric\ Quality$

$Q_g = Gas\ phase\ volumetric\ flow\ rate$

$Q_f = Liquid\ phase\ volumetric\ flow\ rate$

Some of the tests using U-bend and tree orientation of PVC pipe material produced the same peaks as those recorded in literature [3]. The experiments shows that flow regimes influence the relationship between momentum flux fluctuation and excitation force signal, as well as the degree of force excitation. For void fraction tracking, an optical probe is used. [4]

Liu et al. [5] experimented with the slug-churn flow regime, which resulted in force fluctuation due to higher void fraction value. They said that volumetric consistency, or void fraction, has been shown to have an impact on excitation force, but that the effect is more towards the flow regime that in the void fraction itself. Hence, flow regime is an important aspect and must take into preliminarily action when carrying out the FIV analysis.

2.3 Effect of Pipe Geometry

According to Yih and Griffith [3], rectangular piping creates strong transverse vibrations, which are not present in circular piping. This is due to momentum flux; the rectangular pipe has a less value natural frequency and operates close to the resonance region by induced energy fluctuation. According to S. Kim et al. [6], pipe bend curvature has a direct impact on phase separation and void fraction shift. Riverin and Pettigrew [7] reported that the curvature radius at the axial coordinate had marginal momentum flux fluctuation, so no significant impact on the excitation force signal was found.

Furthermore, Riverin and Pettigrew [4] reported parameter such as diameter of the pipe has little impact on the pre-dominant frequency. However, Yi and Griffith's discovery from their experimental study stated that smaller pipe shows significantly greater magnitude of momentum-flux unsteadiness when compared to bigger pipes. Miwa et al. [1] predicted that Yih and Griffith's finding is such because larger piping enhanced two-phase mixing due to the cap bubbles that produced the secondary flow.

The correlations between bubble formation and pipe diameter are discussed in [8] and the critical pipe diameter (D*) is described as follows by Isao and Mamoru.

$$D*_H \equiv \frac{D_H}{\sqrt{\frac{\sigma}{g\Delta\rho}}} \geq 40 \quad (1.2)$$

D_H = hydraulic diameter

σ = liquid phase surface tension

g = gravitational acceleration

$\Delta\rho$ = density difference between two-phase

Their analysis shows that two-phase tubes that surpass the critical diameter fails to form slug bubbles. J. Schlegel et al. [9] concluded that pipes with a value greater than D* should be used to separate flow regimes for larger pipe diameter category.

CHAPTER 3

METHODOLOGY

All subject related to the project including overall flow chart, validation, Altair simulation set up as well as the result of Altair simulation were defined thoroughly in this chapter. In addition, all of the model concept requirements, criteria used, and project analysis input were presented. Finally, this chapter discusses project milestones and Gantt charts for both FYP I and FYP II.

3.1 Flowchart Project

Based on the figure 3.1. below, it shows the plan sequence to tackle the objective study. First and foremost, literature review is an important part of the project study as it is to broaden our mind for the project. In this part of the project, student gather and study various research papers that is related to the topic of the project. This will grow the student's understanding and fundamentals towards the project study. Next, finding the best model set up to run the simulation is crucial in the project sequence. The best model set up can be found by comparing the result of the model set up to the experimental result value of the selected literature review. This process is called validation. Validation is needed to ensure the model set up is correct and accurate before using it for further study. As an output the find out whether the model is correct and accurate, the percentage error of the result must be low. As soon as the model is found correct and accurate, the same model set up will be used for simulation process by using Altair AcuSolve according to the student's own study. Then, the student will tabulate and analyses the data to further discuss the results obtained. Lastly, conclusions and few recommendations for the project study were proposed.

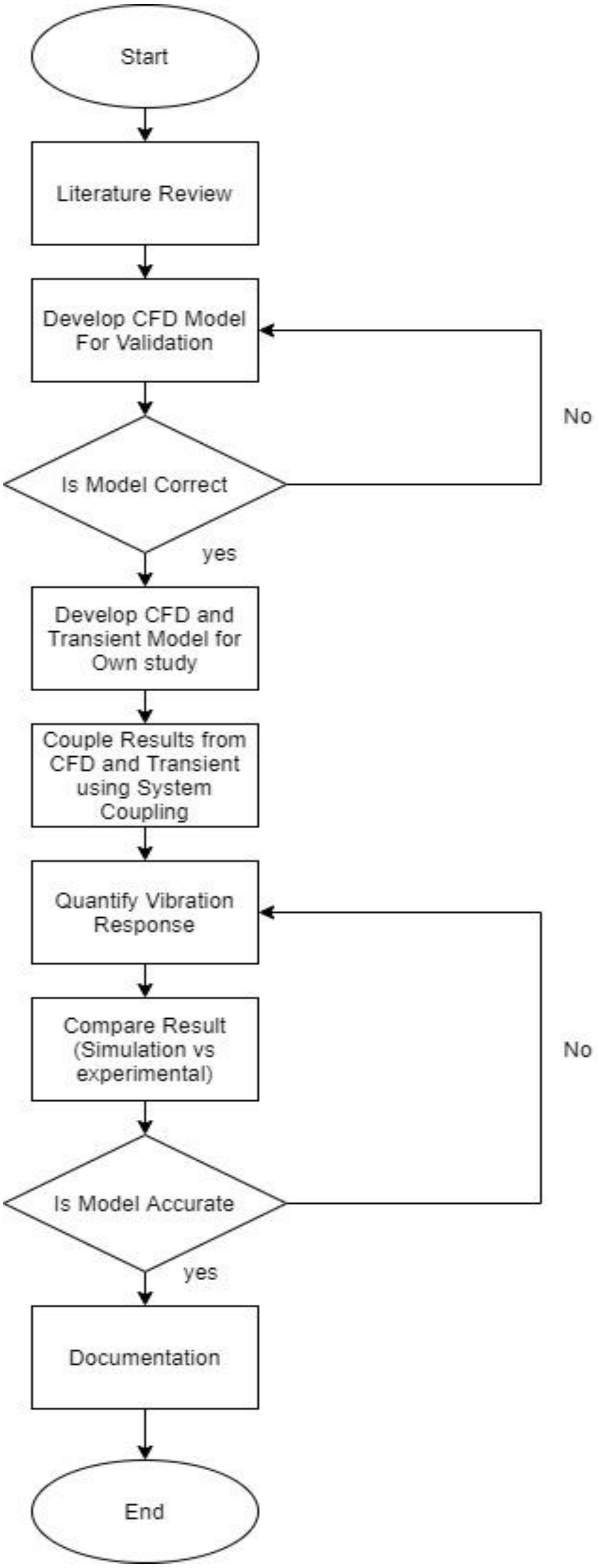


FIGURE 3.1: Project Flow Chart

3.2 Reference Validation Set up

Based on the selected literature for validation, the geometry used is a common combination of vertical and horizontal elbow with upward two-phase flow. It has the dimension of 3 m and 1.9 m for vertical and horizontal pipe respectively as shown in figure 3.2 below. Diameter of the pipe will be 0.0762 m, as well as curvature elbow of 1.5. Based on the literature, 5 flow conditions were used in the simulation. However, considering the time given for the study, only 1 condition with 3 different meshes will be considered in this validation process and that is 10.3 and 0.3 for gas and liquid superficial velocities, respectively. However, having superficial velocities will not be enough to fill in the key component in the boundary conditions tab. Inlet and outlet velocity are needed for the simulation to run. Therefore, equations (3.1) and (3.2) are needed to assume flow rates in the simulations where $V_{inlet\text{-}gas}$ define the gas velocities at the inlet $V_{inlet\text{-}liquid}$ represents velocities of liquid at the inlet. Meanwhile, A_p is the pipe cross-sectional area and A_G and A_L are the gas and liquid injection surface area in the pipe respectively. In terms of mesh, different number of nodes and elements will be developed to represent three types of structured meshes to investigate and compare the accuracy of the meshes with the literature. Instead of using Eulerian Multi Fluid VOF model which were used in the literature, VOF that is offered by Altair AcuSolve will be used. As this simulation will involve two phase which are air and water, both phases will be assumed to be incompressible and no mass transfer between the two phases.

$$V_{inlet-gas} = \frac{V_{SG}\ x\ A_P}{A_g} \tag{2.1}$$

$$V_{inlet-liquid} = \frac{V_{SL}\ x\ A_P}{A_L} \tag{3.2}$$

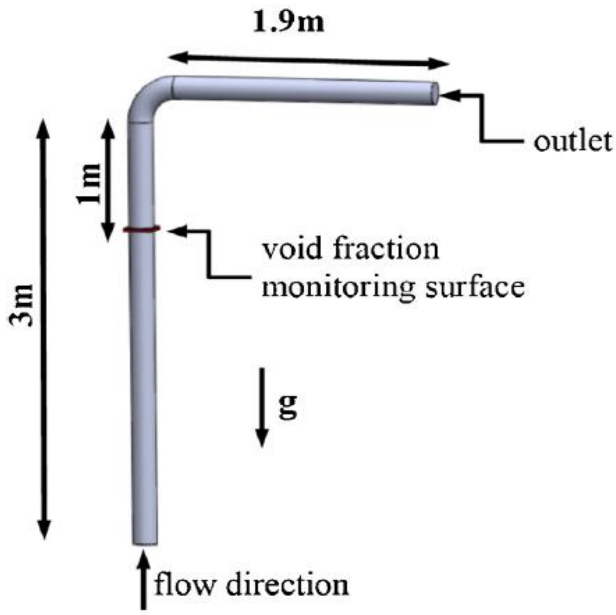


FIGURE 3.2: Validation Geometry from Literature [10]

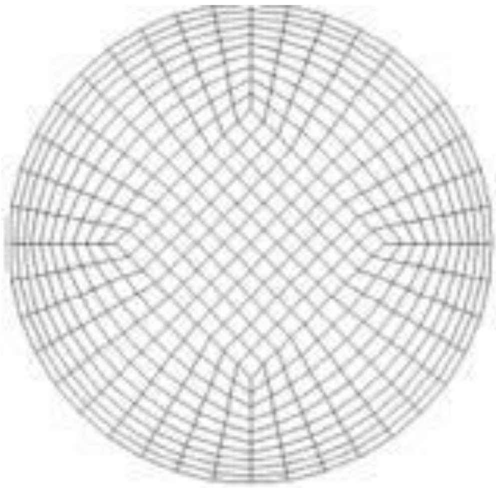


FIGURE 3.3: Validation Mesh from Literature [10]

TABLE 3.1: Under Relaxation Factors [10]

Parameter	Under-Relaxation Factors
Pressure	0.3
Density	1
Body Forces	1
Momentum	0.7
Volume Fraction	0.8
Turbulent Kinetic Energy	0.8
Specific Dissipation Rate	0.8
Turbulent Viscosity	1

As for validation, results are needed to compare and validate if the model set up is correct and accurate to be used in own simulation study. Therefore, in this validation, results will be compared in terms of mean void fraction of the two-phase flow. Void fraction in a two-phase flow is defined as the fraction of the channelled volume that is occupied by the gas phase. The result from the literature that will be used to validate is the mean void fraction of 0.72.

3.3 Model Geometry for Validation

As for model geometry, a vertical straight pipe with 4-meter in length and 0.074-meter in diameter shown in figure 3.4 has been prepared to validate with the model in literature. The geometry model has been divided into several parts which has 5 volume parts and 16 pipe surfaces as described in figure 3.5. This has to be done to allow the formation of the specific flow regime. This process is called perturbation.

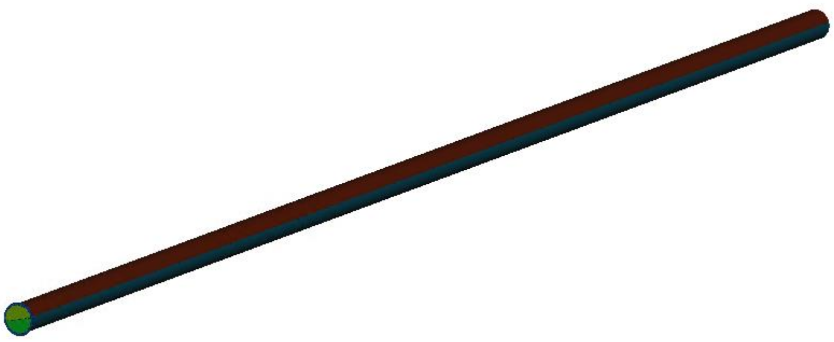


FIGURE 3.4: Model Geometry for Validation

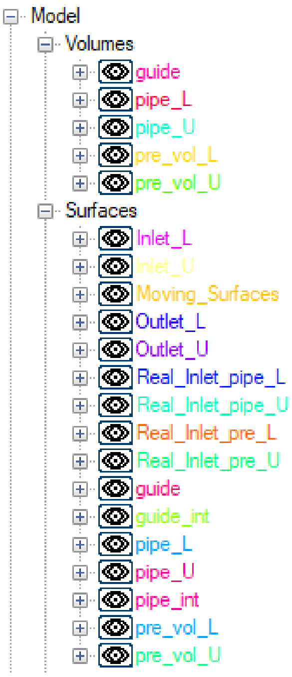


FIGURE 3.5: Perturbation Set up

3.4 Boundary Condition for Validation

For boundary conditions, velocity has been set with the value of 10.3 m/s and 0.3 m/s for gas and liquid respectively in order to have the same result as literature. Moreover, the simulation has been set up so that gravity of 9.81 kg/m^3 will have impact on the result. Density of water, air and Perspex is as shown in table 3.2. Furthermore, Surface tension between air and water is 0.074. This value is decided based on atmospheric temperature. In addition, pressure of 101.3 KPa has been set at both Upper and lower outlet.

TABLE 3.2: Density of Air, water, and Perspex

Material	Density
Air	1.185 Kg/m^3
Water	998.6 Kg/m^3
Perspex	1180 Kg/m^3

3.5 Meshing for Validation

In terms of mesh, meshing is essential for simulation’s computer-aided engineering (CAE) operation. The accuracy, speed, and convergence of the results are all determined by the mesh. The time it takes to create and mesh a model is also a significant part of the time it takes to get results from CAE solutions. As a result, the more efficient and better the meshing methods are, the better the solution would be in the end. The smoother the mesh of the geometry is compared to a coarse mesh, the longer it takes for the program to produce the same result. In this validation process, only one type of mesh consists of 468,193 number of nodes and 923,358 number of elements were used in this study as shown in figure 3.6.

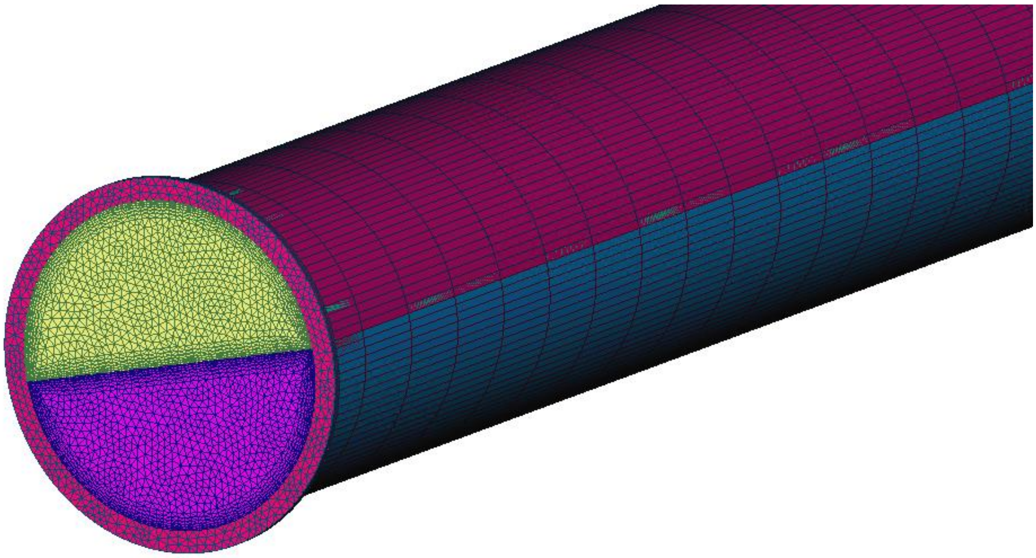


FIGURE 3.6: Structured Mesh

3.5 Materials Piping for Validation

The translucent Perspex piping material used in the experimental setup has outstanding power, optical consistency, and stiffness, making it simpler to fabricate the piping model and giving a direct picture of the water-air transmission in the pipe. The Perspex also has good resiliency, elasticity, and the potential to resist friction. Perspex pipe has a melting point of 160°C and can tolerate the temperature of air and water. It has a density of 1180 kg/m^3. The type of fluids implemented in this simulation are air and water with densities of 1.225 kg/m^3 and 997 kg/m^3 respectively.

3.6 Level Set Approach

In this simulation, 2 different level set approach were setup and to be compared. First approach is by using conventional Level set as provided by Altair AcuSolve and another is Level set with Back-and-Forth Error Compensation and Correction (BFECC) approach. Both Level set approaches rely on two additional staggers to track the interface. As for the conventional Level set approach, the first stagger governs the transport of the interface, and the second stagger controls the sharpness of the interface. On the other hand, BFECC Level set method includes extra stagger iterations to reduce the amount of diffusion in the solution field. This method captures multiphase interfaces more sharply than the conventional level set approach. However, extra stagger iteration will lead to more computing time compared to standard level set approach. Therefore, 2 different results will be compared to the results in the literature.

3.7 Key Milestones
3.7.1 Final year Project 1 (FYP 1)

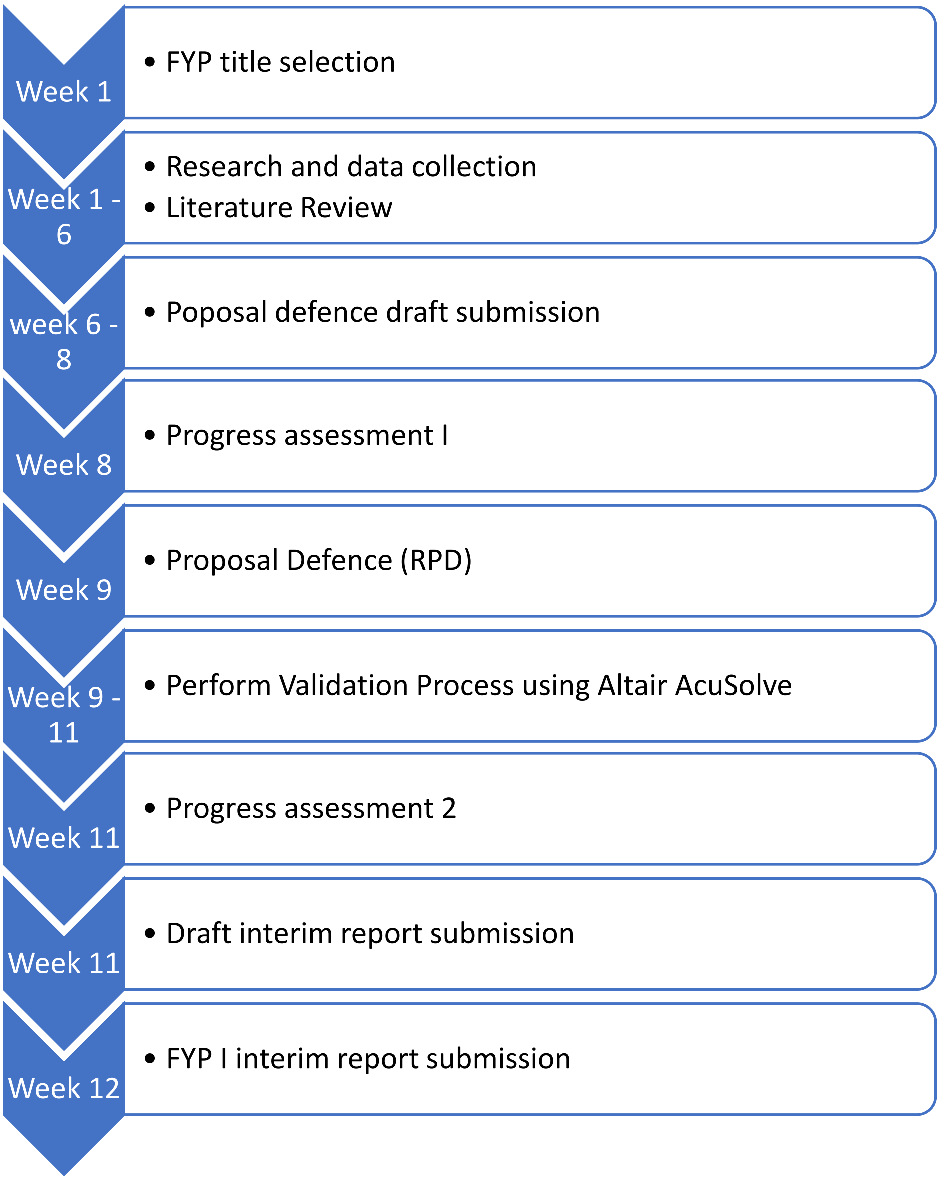


FIGURE 3.7: FYP I Milestone

3.7.2 Final year Project II (FYP I1)

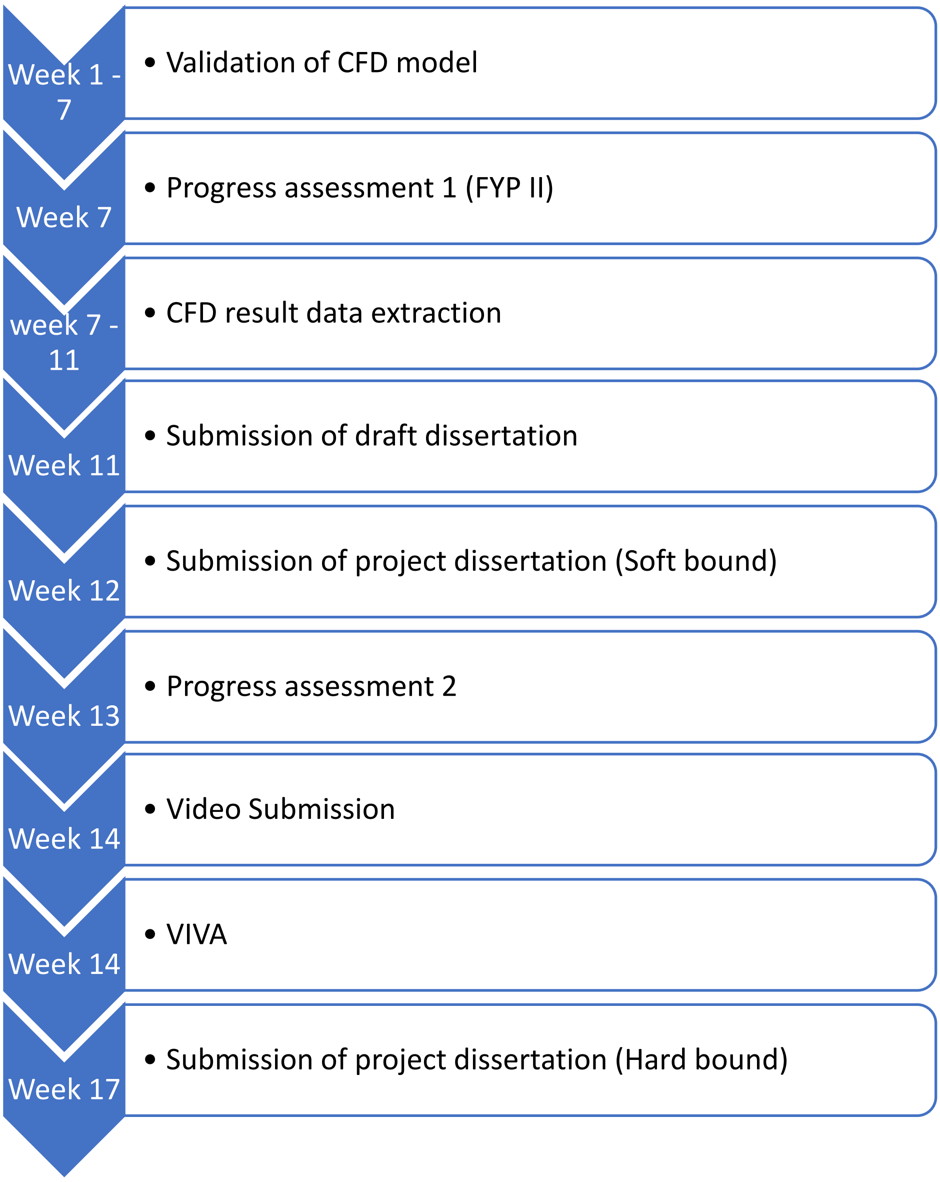


FIGURE 3.8: FYP II Milestone

TABLE 1.3: Gantt Chart for FYP 1

Activity	Week											
	1	2	3	4	5	6	7	8	9	10	11	12
Introduction to title	■											
Research and data gathering	■	■	■	■	■	■						
Submission of draft proposal defence						■	■	■				
Progress assessment 1								■				
Proposal Defence									■			
CFD validation									■	■	■	
Progress Assessment 2											■	
Submission of draft interim report											■	
Submission of interim report												■

TABLE 3.4: Gantt Chart for FYP 2

Activity	Week																
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17
Validation and simulation of CFD																	
Submission of progress assessment 1																	
Flow behaviour analysis of piping system																	
Draft dissertation submission																	
Project dissertation submission (Soft bound)																	
Progress Assessment 2																	
VIVA																	
Project dissertation submission (Hard Bound)																	30-Apr

CHAPTER 4

RESULT AND DISCUSSION

4.0 Overview

This chapter emphasizes the results and discussion that have been obtained from the validation discussed in chapter 3. In this chapter, the results were focused on the validation model to discuss the result whether the model is correct and accurate before using it for further research model. The result that will be discussed in this chapter is the result output from the CFD simulation of the vertical straight pipe model geometry as stated in chapter 3.

4.1 Convergence Study

Figure 4.1 and 4.2 displays the residual ratio of conventional level set approach and BFECC approach, respectively. The simulation for this convergence study has been carried out on the vertical straight pipe as stated in chapter 3 with the same setup as mentioned in the chapter as well. This convergence study was generally performed to determine whether the result is stable to be used in the study. In this study, residual ratio method was performed in convergence check. The norm of the residual is normalized with respect to the norm of the forces making up the residual. This ratio is relevant measure, because it measures the ratio of the out-of-balance forces to the value of the forces. However, to check the convergence, all residual ratios must fall below the specified tolerance in the setup. In this case, the specified tolerance was 0.01.

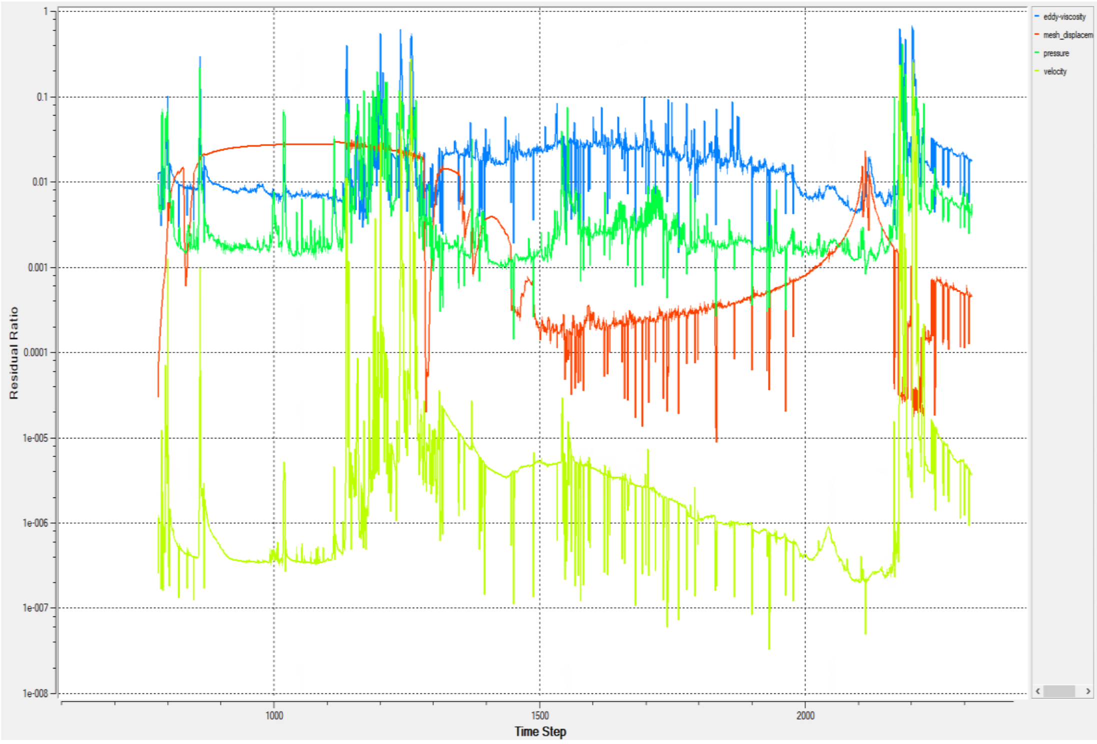


FIGURE 4.1: Residual Ratio for Standard Level set approach

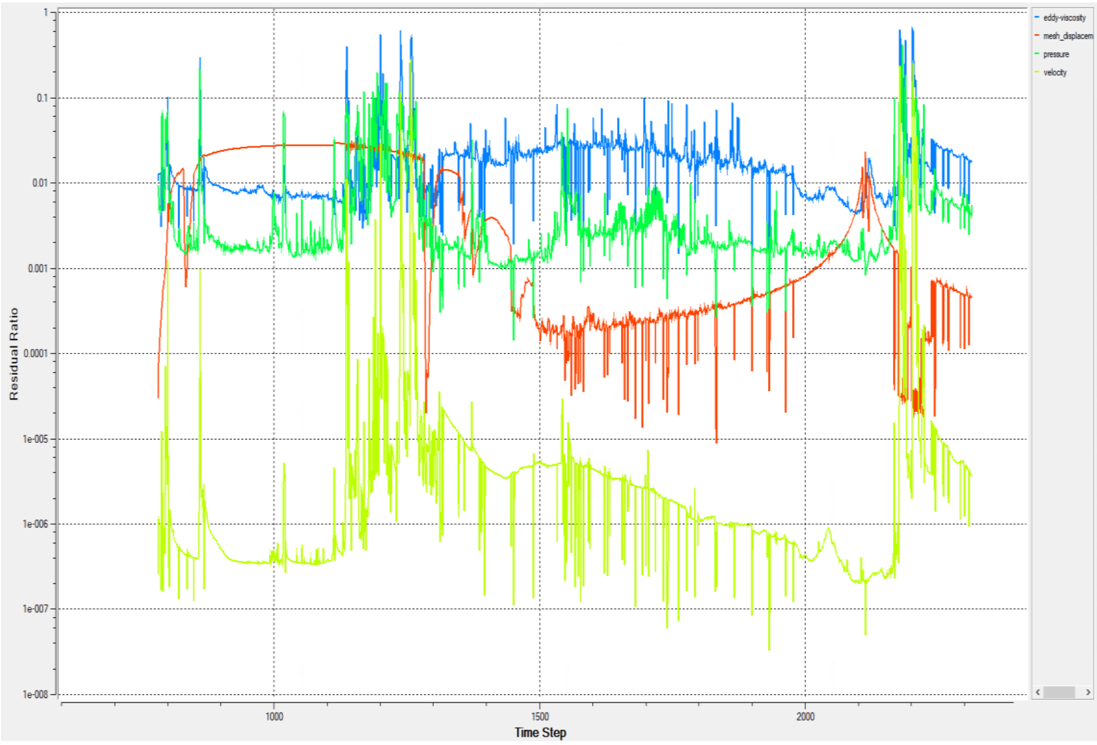


FIGURE 4.2: Residual Ratio for BFECC approach

In the case of the validation result, both levelset approaches exceeds the value of specified tolerance which means they have low convergence accuracy. However, by comparing between both results, BFECC gives much better convergence than the standard levelset approach. Standard approach has multiple spike that exceeds the value of 0.1 whereas BFECC has 2 spikes that exceeds 0.1. Therefore, BFECC approach will be considered as acceptable. However, both results produced form both approaches will be discussed and compared in further discussion.

4.2 Flow Regime Identification

In this simulation, the conditions used were 10.3 m/s and 0.3 m/s for gas and liquid, respectively. To predict the flow regime, baker’s chart will be used as shown in figure 4.3. By plotting the specified conditions of superficial gas and liquid velocity, the point falls under churn flow regime.

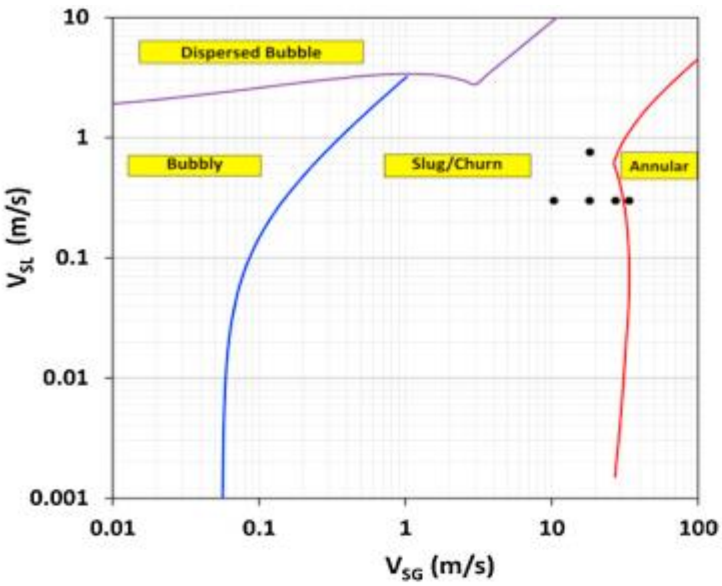
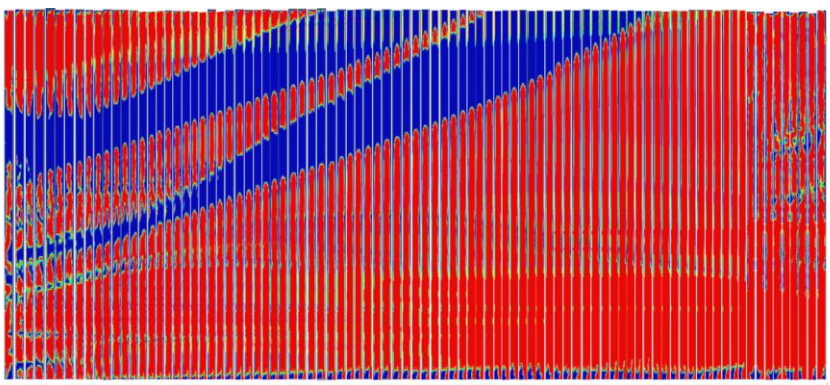


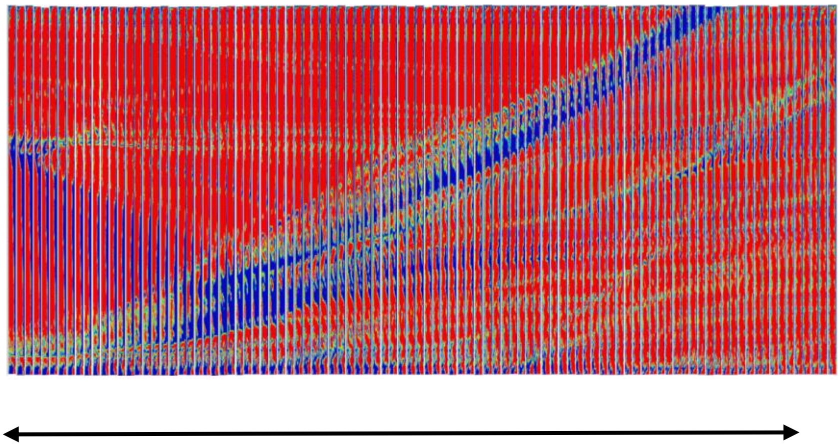
FIGURE 4.3: Baker's Chart for Vertical Pipe [10]

4.3 Time evolution of the flow

The time flow evolution of the water and air contour of the vertical pipe based on air volume fraction of the validation model for both standard level set and BFECC approaches have been presented in table 4.1. Both results were taken from 1.26 seconds to 2.31 seconds. In the case of standard level set approach, the generation of the first wave occurs earlier than in BFECC approach. The wave slowly propagates and grow to slug. BFECC approach on the other hand as shown in the table. The flow first wave is later than standard approach which is at 1.50 seconds. However, by comparing flow regimes as stated in the literature, the flow should be in churn flow. This means that only BFECC generate churn flow whereas standard approach failed to generate churn.

TABLE 4.1: Time flow Evolution of the water and air contour of the vertical pipe based on air volume fraction.

Approach	
Conventional	 1.26 seconds to 2.31 seconds

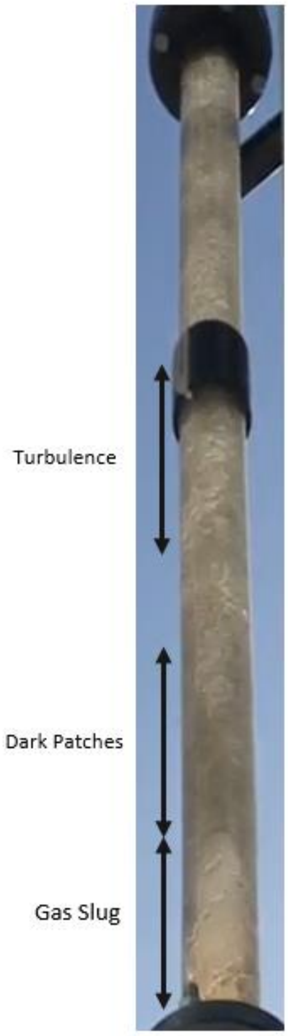
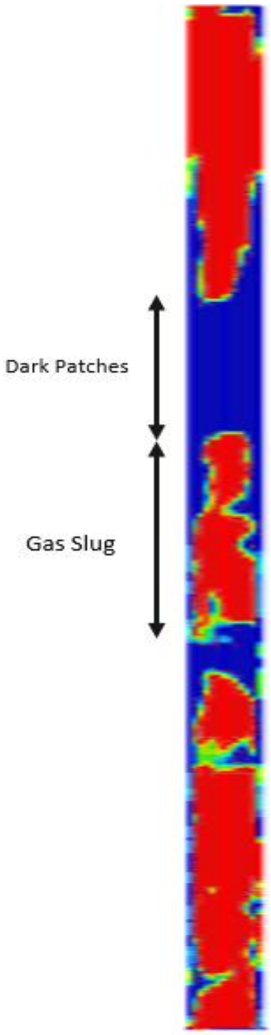
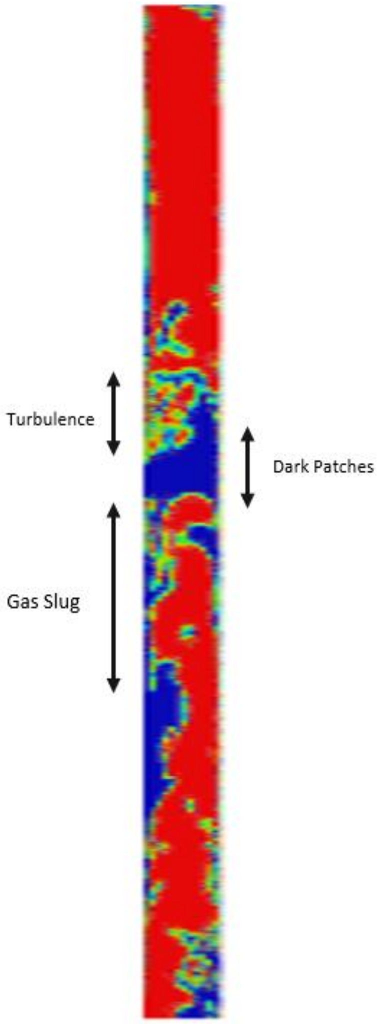
Back and Forth Compensation and correction	 1.26 seconds to 2.31 seconds

4.4 Comparison Literature and Validation Model with Both Approaches

In order to support the validation model, few samples were taken from the literature as a reference for comparison. The samples were recorded and observed by the author in the literature during experiments. Then, several samples were also taken from the validation simulation of both level set approaches. The main purpose is to discover if the CFD can regenerate the same behaviour of the real experiments.

The samples from the literature [10] and samples from CFD simulation is then put side by side to see the similarities of the flow behaviour. The first set of comparison is presented in table 4.2. The first sample of the experiment from the literature displays the cyclic liquid. In the sample picture, dark patches are the liquid structures. On the other hand, clear patches illustrated the gas structures. From the experiment perspective, it appears that the gas slug flows upwards with the nose shape of an inverted U-shape before the cyclic liquid. Then comes the turbulency of the flow happens after a section of dark patches. This occurrence then be compared to the CFD simulations of both approaches to see if they produce the same event. However, BFECC approaches produced better similarities than Conventional levelset approach. The conventional methods failed to form a turbulency of the flow after the section of liquid patches whereas BFECC method able to produce the same event.

TABLE 4.2: Comparison 1 of experimental data VS Conventional and BFECC Level set

<table>
<tr><th colspan="3">Comparison 1</th></tr>
<tr><th>Literature [10]</th><th>Conventional level set</th><th>BFECC level set</th></tr>
<tr>
<td>
</td>
<td>
</td>
<td>
</td>
</tr>
</table>

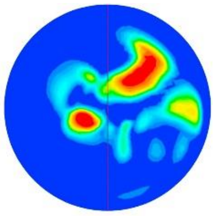
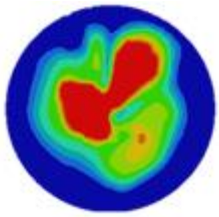
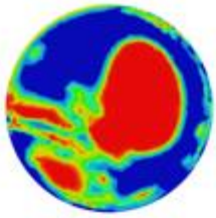
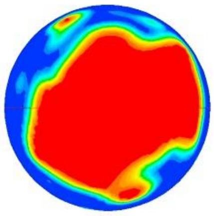
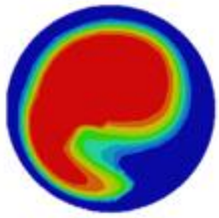
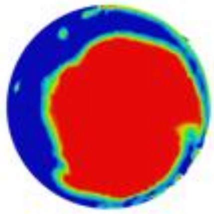
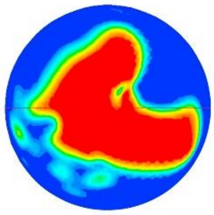
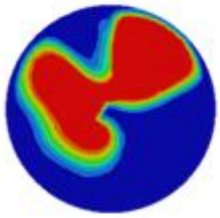
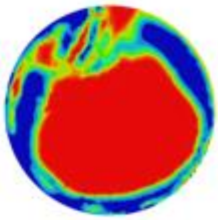
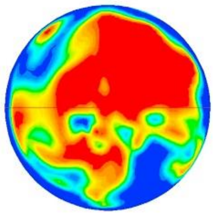
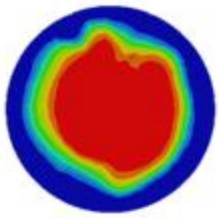
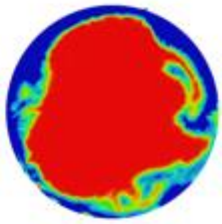
Moreover, second comparison is figured in table 4.3. This comparison is based on void fraction at the same cross section for both literature and CFD. The blue colour indicates the liquid phase, and white and red indicate the gas phase for both WMS and CFD of both level set approaches, respectively. In this comparison, main criteria to be considered is the bubble formation as shown in experimental data gained from the literature. The experimental data on the first row illustrates that the area is covered by water phase and an irregular shaped of gas phase at the centre along with few bubbleson the side. Unfortunately, all other simulation samples failed to show exactly as shown in the experimental samples. However, the closest sample among the 3 simulation samples is the sample of BFECC level set. It is observed that, the shape follows of the experimental samples with an irregular shaped gas phase at the centre with water phase surrounding it. Formation of bubble was also presence in the BFECC sample.

As for the second row, the experiment samples describes that the area is almost fully covered by gas phase at the centre and liquid phase at the wall of the pipe section along with less amount of bubble formation. In this case, all other simulation samples produce good results. However, sample of conventional level set has failed to form a single bubble as described for experiment sample.

Comparison in third row has the most differences between experiment sample with other simulation samples. The experiment sample illustrates that the area is filled with a face with horns shape and a single drop of water as the eye. However, among all other 3 simulation samples, BFECC level set has the closest result as it has the formation of horns.

On the final row, the experiment sample shows mass formation bubble with liquid phase at the centre of the section. However, BFECC produced the best similarities among other simulation sample. The water phase on the right side of the cross section of the BFECC sample seem to form a ‘bay’ towards the centre of the area, which is quite similar to the experimental data. Furthermore, mass bubble formation is also there in the sample result.

TABLE 4.3: Sample of Comparison of Cross Section between Published Experiment VS Published Simulation VS Conventional Level set Validation Model VS BFECC Validation Model

Cross Section			
Experiment (Published Data)	Simulation (Published Data)	Conventional Level Set	BFECC Level Set
			
			
			
			

CHAPTER 5
CONCLUSIONS AND RECOMMENDATIONS

5.1 Conclusion

The multiphase flow of gas and liquid contained in the refining and petrochemical industries exhibits many forms of vibration in terms of frequencies and vibration amplitudes due to the influence of many factors including topology, superficial velocities of fluids and the void fraction. As a result of these factors, the reliability and performance of the systems are reduced, and the systems can malfunction at times. As a result, a thorough examination of the influence of topologies that lead to different vibration cases of pipe conveying fluids is important. The validation model was used to simulate the two fluids, for example, air and water, in order to analyse the formation of the slug flow in the pipe conveying fluids and was compared to the literature [10]. Furthermore, different level set approaches were analysed in this research. Both approaches give different results. However, BFECC is more reliable than the conventional approach. Despite all that, it is yet to find solutions for few outputs such as vibrational behaviour and area-weighted cross sectional void fraction due to difficulties in the setup. The setup was done by using Altair Hyperview to analyse the void fraction of the cross-sectional pipe. Many efforts being done trying to obtain the result. Unfortunately, due to time constraint, the proper configurations could not be completed at this time. As a result, vibrational analysis was not able to proceed due to lack of data from the void fraction.

5.2 Recommendations

Throughout the study, the author has given his efforts trying to figure out on how to obtain the best results from the simulation. Therefore, the author highly recommended for the next person who will be continuing this project to increase the final stop time for the vertical pipe. This will produce enough results to move on to the next step. Furthermore, extra literature is highly recommended as to increase understanding when viewing the results.

REFERENCES

[1] S. Miwa, M. Mori, and T. Hibiki, “Two-phase flow induced vibration in piping systems,” *Prog. Nucl. Energy*, vol. 78, pp. 270–284, Jan. 2015, doi: 10.1016/j.pnucene.2014.10.003.

[2] T. Fujikawa *et al.*, *Flow-Induced Vibrations*, Edition 1. Hungary: Elsevier Ltd, 2008.

[3] T. S. Yih and P. Griffith, “Unsteady Momentum Fluxes in Two-Phase Flow and the Vibration of Nuclear Reactor Components,” Massachusetts, 1968.

[4] J. L. Riverin, E. de Langre, and M. J. Pettigrew, “Fluctuating forces caused by internal two-phase flow on bends and tees,” *J. Sound Vib.*, vol. 298, no. 4–5, pp. 1088–1098, Dec. 2006, doi: 10.1016/j.jsv.2006.06.039.

[5] Y. Liu *et al.*, “Experimental study of internal two-phase flow induced fluctuating force on a 90° elbow,” *Chem. Eng. Sci.*, vol. 76, pp. 173–187, Jul. 31 2012, doi: 10.1016/j.ces.2012.04.021.

[6] S. Kim, J. H. Park, G. Kojasoy, J. M. Kelly, and S. O. Marshall, “Geometric effects of 90-degree Elbow in the development of interfacial structures in horizontal bubbly flow,” *Nucl. Eng. Des.*, vol. 237, no. 20–21, pp. 2105–2113, Nov. 2007, doi: 10.1016/j.nucengdes.2007.02.007.

[7] J.-L. Riverin and M. J. Pettigrew, “Vibration Excitation Forces Due to Two-Phase Flow in Piping Elements,” *J. Press. Vessel Technol.*, vol. 129, no. 1, pp. 7–13, Feb. 2007, doi: 10.1115/1.2388994.

[8] K. Isao and I. Mamoru, “Drift flux model for large diameter pipe and new correlation for pool void fraction,” *Int. J. Heat Mass Transf.*, vol. 30, no. 9, pp. 1927–1939, Sep. 1987, doi: 10.1016/0017-9310(87)90251-1.

[9] J. P. Schlegel, S. Miwa, S. Chen, T. Hibiki, and M. Ishii, “Experimental study of two-phase flow structure in large diameter pipes,” *Exp. Therm. Fluid Sci.*, vol. 41, no. 2012, pp. 12–22, Sep. 2012, doi: 10.1016/j.expthermflusci.2012.01.03.

[10] *Parsi, M., et al. (2016). "Assessment of a hybrid CFD model for simulation of complex vertical upward gas–liquid churn flow." Chemical Engineering Research and Design 105: 71-84*

[11] *Giraudeau, M., Mureithi, N. and Pettigrew, M., 2020. Two-Phase Flow-Induced Forces on Piping In Vertical Upward Flow: Excitation Mechanisms And Correlation Models.*

[12] F. A. Holland and R. Bragg, Fluid flow for chemical engineers. London: Edward Arnold, 1995.

[13] Xue, Y., et al. (2016). "Investigation on the void fraction of gas–liquid two-phase flows in vertically-downward pipes." International Communications in Heat and Mass Transfer 77: 1-8.

[14] Pao, W., Sam, B., Nasif, M., & Norpiah, R. (2017). Numerical Validation of Two-
Phase Slug Flow and its Liquid Holdup Correlation in Horizontal Pipeline. Key Engineering Materials, 740, 173-182.

[15] S. Ban, W. Pao, and M. S. Nasif, “Numerical simulation of two-phase flow regime in horizontal pipeline and its validation,” International Journal of Numerical Methods for Heat & Fluid Flow, vol. 28, no. 6, pp. 1279–1314, 2018.