Computational Fluid Dynamics Simulation of Malaysian Waxy Crude Oil-water Two-phase Flow Based on Contact Angle in Horizontal Pipe

Geerbasini Elangovan
School of Engineering
Asia Pacific University of Technology and Innovation (APU)
Kuala Lumpur, Malaysia
tp056169@mail.apu.edu.my

Muhammad Syahmi Mokhtar Yazid
School of Engineering
Asia Pacific University of Technology and Innovation (APU)
Kuala Lumpur, Malaysia
syahmi.afif@apu.edu.my

Juhairi Aris Bin Muhamad Shuhili
School of Engineering
Asia Pacific University of Technology and Innovation (APU)
Kuala Lumpur, Malaysia
juhairi.shuhili@apu.edu.my

Rosli Yusop
School of Engineering
Asia Pacific University of Technology and Innovation (APU)
Kuala Lumpur, Malaysia
rosli.yusop@apu.edu.my

Lee Jang Hyun
Petroleum Engineering Department
Universiti Teknologi PETRONAS
Perak Darul Ridzuan, Malaysia
lee.janghyun@utp.edu.my

Adeline Sneha John Chrisastum
School of Computing
Asia Pacific University of Technology and Innovation (APU)
Kuala Lumpur, Malaysia
adeline.john@apu.edu.my

Muhammad Adidinizar Zia Ahmad Kusairee
Department of Business and Management
Universiti Teknologi Mara Perak Branch Tapah Campus,
Malaysia Perak Darul Ridzuan, Malaysia
adidi627@uitm.edu.my

***Abstract* — The petroleum sector frequently uses pipelines to transport crude oil. Due to varying topography and climatic conditions, the characteristics of crude oil from various sources varied, changing the transport profile while activities were underway. The computational fluid dynamics model for this investigation was created using computer code. The governing equations were used to analyze how different aspects of crude oil, such as density and viscosity affected the transportation profile. The main objective of this study was to examine the CFD Fluent simulation of the flow pattern, pressure drop, and liquid holdup of Malaysian waxy crude oil-water. A 5.08 m pipe design using Autodesk Inventor and the CFD Fluent was loaded with the pipe's geometry. The developed model was validated by comparing the simulation findings to those from the experimental work. To create a base model for the Malaysian waxy crude oil-water, a mesh independence analysis was done to identify the best kind of mesh for the specified geometry. The type of mesh was decided after a few experiments. The efficiency of the meshes generated by the various types of mesh was evaluated using the ratio of the degree of skewness to the total number of meshes. By choosing the mesh and turbulence CFD that are most suitable for the waxy crude oil and water, the ANSYS Fluent software will be used to study the many options and features of the software and develop validated models.**

Keywords*— *Computational Fluid Dynamics (CFD), Menter's Shear Stress Transport (SST), Renormalization Group (RNG), Reynolds Stress Model (RSM), Wax Appearance Temperature (WAT)

I. Introduction

The fast development of technologies to generate oil and gas in commercial amounts had been prompted by the rising need for energy from petroleum sources. In the petroleum sector, pipelines are frequently utilized for the transportation of crude oil and natural gas from the reservoir, through the wellhead, to the location where it will be stored (Sanni, Olawale, & Adefila, 2015; Al-Waily, Al-Baghdadi, & Resan, 2017). Turbulence is present when fluid e.g., crude oil is flowing in an unsteady manner via a pipe. The Reynolds number can be used to determine how turbulent this flow was on average. Analysis of fluid flow behavior has previously been studied from both an experimental and theoretical perspective. It proved to be capital-intensive, necessitating an effective, affordable, and trustworthy computational tools (Al-Waily, Al-Baghdadi, & Resan, 2017)

Multiphase flow consequently happened commonly in oil wells. In a pipeline, the flow regime and corresponding flow rate were often used to examine multiphase flow. Internal corrosion rate would be impacted by multiphase flow, which was significantly different from single-phase flow in a pipeline in terms of corrosion due to differing hydrodynamic s and associated turbulence (Hernández & Vorobieff, 2020). The k- ε turbulence model was used to examine the effects of eccentricity on friction factor and holdup for both laminar and turbulent scenarios. It had been discovered that eccentricity raises the friction factor. The model outperformed the K- ω turbulence model in terms of predicting wavelength and pressure distribution (Huang, Sun, Zhang, Li, & Wei, 2019; Lee, Heo, Sohn, & Ko, 2021). Core annular down flow in FLUENT was modelled using the Eulerian-Eulerian based VOF technique (Ghosh & Noboru, 1991).

A. *Literature Review*

When two separate phases interacted and flew through a channel with one another, each phase is thought of as a mass or volume of matter. This was referred to as two-phase flow network layers (Awad, 2012). Combinations of solid, gas, and/or liquid phases could exist in the two phases. Although multiphase flow containing three phases was occasionally used in engineering, most multiphase applications include two phases (Faghri & Zhang, 2006). Nearly all forms of flow patterns that one might see during the flow of a model oil and water in horizontal pipes, had maybe one of the most complete flow pattern classifications as shown in Figure 1 (Elseth, 2001)

In addition to having substantial academic significance, the scientific description of the properties of the horizontal oil-water two-phase flow pattern also had practical implications for overcoming challenging technological problems in the petroleum industry (Lin, Kew, & Cornwell, 2001). Due to many factors, including flow velocity and phase-separated holdup, the oil-water two-phase flow generated a variety of flow patterns in the mixed flow. Contrary to single-phase flow, two-phase flow had the characteristics of nonlinearity and interphase slip which made it more challenging to identify flow patterns and detect flow pattern conversion (Tomar, 2015).

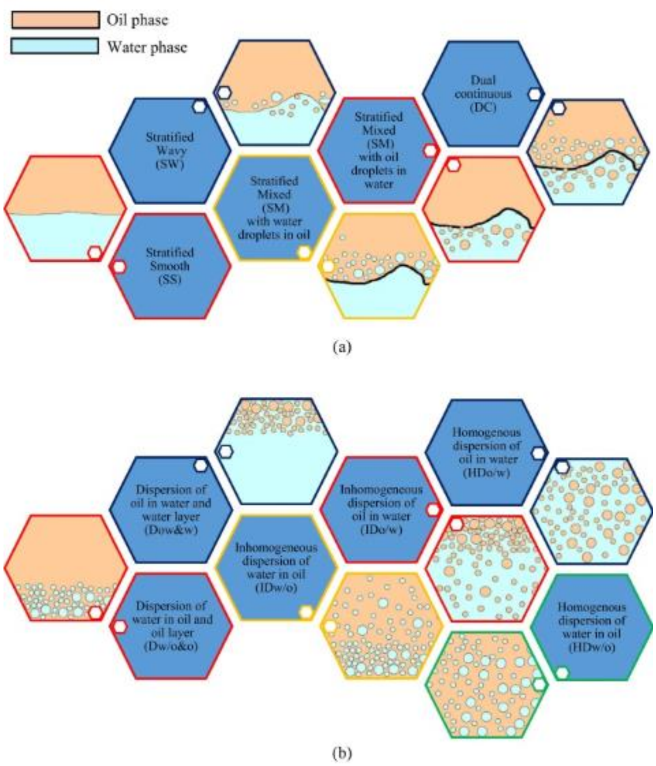


Figure 1. Oil-Water flow pattern (a) segregated flow and (b) dispersed flowFig. 1. Oil-Water flow pattern (a) segregated flow and (b) dispersed flow (Elseth, 2001)

II. METHODS

A. *Mesh Selection Process Flow*

The entire procedure for creating the software model in Ansys Fluent was depicted in the flowchart in the Figure 2. Pre-processing, processing, and post-processing were the three processes that make up the simulation process. As seen, each stage is connected to the others. For instance, the proper mesh selection and boundary condition setting might lead to convergence of a better outcome. In the pre-processing step, in addition to the various mesh types that could be used for the model, such as tetra and quads, the definition of the design, meshing, modelling geometry, and simulation setup take place. The mesh size and mesh study are chosen independently to produce accurate findings.

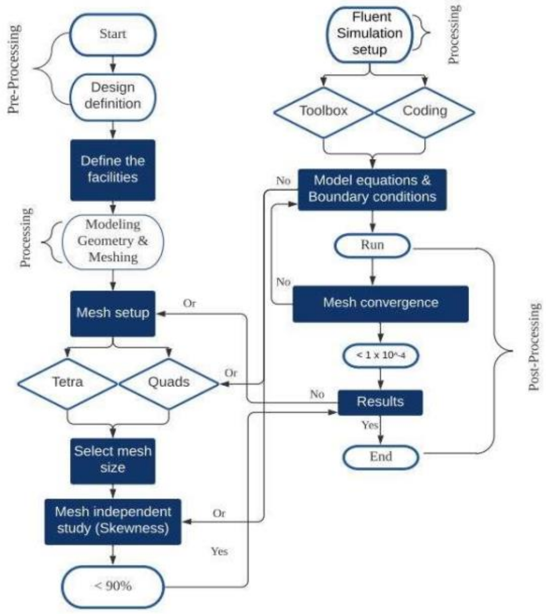


Figure 2. Mesh Selection Procedure

B. *Facility Layout and Fluid Properties*

The experimental campaign began by describing the test fluids for the research, as shown in the table below. Waxy Terengganu crude oil from Malaysia and filtered water were used as test fluids throughout the entire testing process. The waxy crude oil used for the research was categorized as mild waxy crude oil because of its low WAT and low pour point of 25°C and 18°C, respectively. Filtered water was chosen to be fed into the facility to ensure that only liquid-liquid flow took place in the system.

TABLE 1. Inputs for Mesh Selection

Properties	Water	Crude Oil
API gravity	-	41.4
Viscosity (cp)	1.0	1.75
Density (kg/m3)	1000	818
Flash point (°C)	-	<19
Pour point (°C)	-	18
Wax content, wt%	-	16.15

WAT (℃)	-	26
Retention time, (min)	-	<1
Asphaltenes, wt%	-	0.06

At the Malaysia Petroleum Resources Corporation Institute for Oil and Gas (UTM-MPRC for Oil and Gas) of Universiti Teknologi Malaysia, the experimental work was done (UTM). In the Figure 3, the full experimental setup was shown. The experimental setup, which had an inner pipe diameter of 50.8 mm and a total length of 49.27 m, was created as a closed-loop facility. The 3.0 m test section, which provided as a visual representation of the flow patterns and allowed for their observation, was constructed from a transparent acrylic pipe.

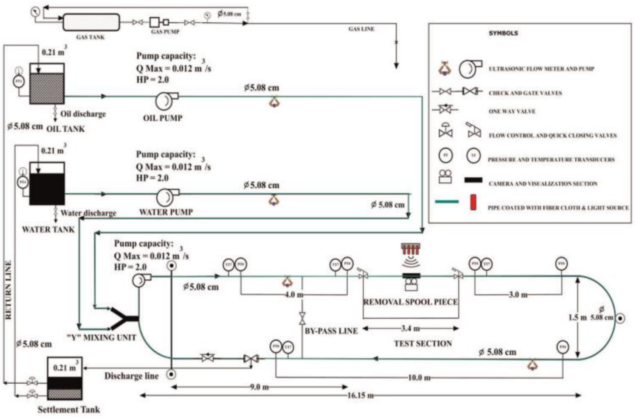


Figure 3. Schematic diagram of oil-water test facility

The experimental parameters served as the foundation for the simulation model's construction. The steps that needed to be finished before releasing the results. However, the geometry-based design of the pipe was the initial step of all. To visualize the experiment pipe's actual characteristics, Autodesk Inventor was employed. In addition, ANSYS Fluent would import the displayed model for mesh production. Inventor provided a better interface and made it easier to generate models than ANSYS Fluent. The pipe geometry produced by Autodesk is shown in Figure 4.

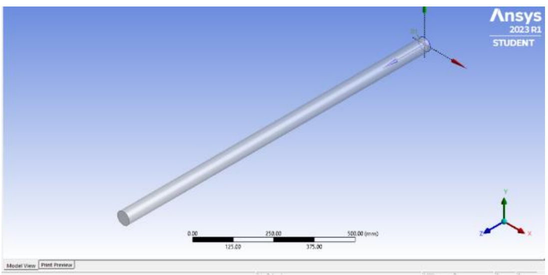


Figure 4: Pipe geometry in Autodesk

It is necessary to import the model geometry into ANSYS. fluent to begin with. the first stage of simulation, mesh generation. the choice of inflation and mesh dimensions. The mesh sizing process's quality of size was correctly chosen, with closeness and curvature defined as the importance center. Additionally, skewness number must be set in the mesh independence study to assess the mesh's quality.

The pressure-based, transient time, and absolute velocity formulations were often set as a first stage in choices. The gravitational acceleration for the Y-axis was then adjusted to -9.81 m/s2 as shown in Figure 5. After that, as the model was simulating both air and oil, the multiphase option was picked, and the volume of fluid (VOF) approach with an explicit scheme and two numbers of Eulerian phases was chosen.

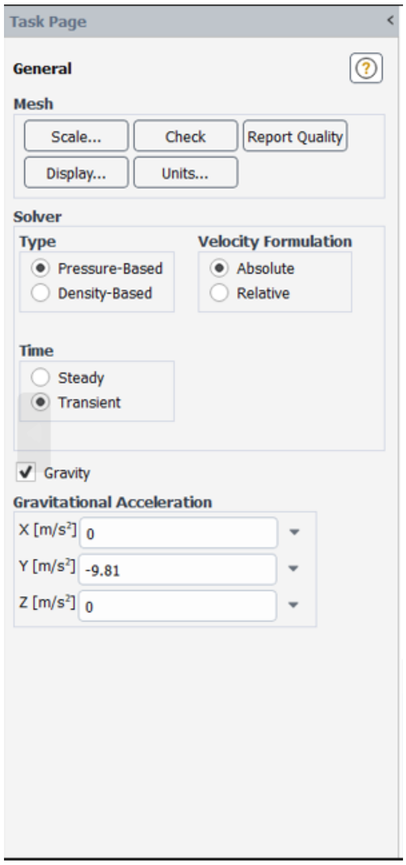


Figure 5: Task Page for Mesh Convergence

III. RESULT AND DISCUSSION

The pressure drop, flow pattern and liquid holdup in a horizontal pipe was examined and then the results were compared with the experimental data of Malaysian waxy crude oil-water flow. The flowchart that contained the process followed to complete this project was shown. The use of flowcharts was crucial when creating and planning a process. At the Malaysia Petroleum Resources Corporation Institute for Oil and Gas (UTM-MPRC for Oil and Gas), Universiti Teknologi Malaysia (UTM), all experiments and measurements for the flow of waxy crude oil and water in pipelines in Malaysia were conducted. Figure 6 depicted the full experimental setup.

The ANSYS Fluent program would be used to select the top turbulence model for the two-phase flow under test since different turbulence models yield different simulation outcomes. To produce an accurate output, the best mesh was selected. The results of the various scenarios were compared to those of the experiment to guarantee that each result is accurate. The pipe had been instrumented to enable continuous temperature and pressure drop monitoring. The internal diameter (ID) was 5.08 cm. The test segment was equipped with thermocouples to track the pipeline's thermal state continually. To reduce heat loss, fibre cloth was wrapped over the test section's exterior pipeline up to the settling tank. As seen in Figure 7, the closed loop was additionally coupled to the 'Y' mixing section at the entry.

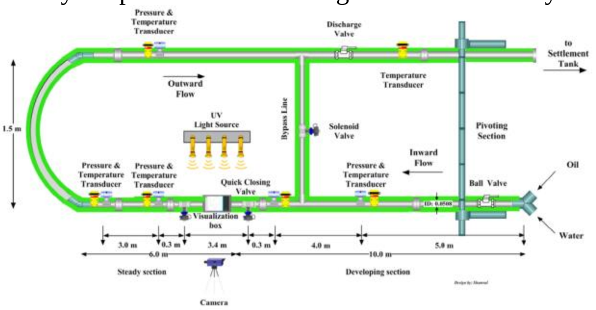


Figure 6: Schematic diagram of multiphase flow loop test section

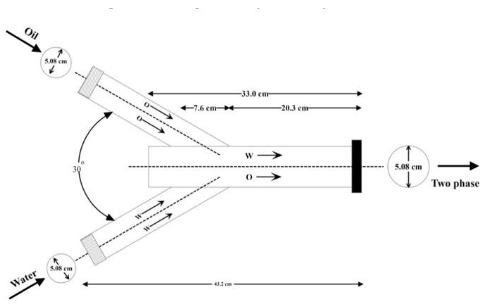


Figure 7: Y Section

A 3D pipe geometry had been made to roughly represent the horizontal pipe's actual cross-sectional area, length, and inner diameter. Autodesk Inventor was utilized to define pipe geometry when designing it since it produced good interfaces. The next stage was selecting the mesh, and four distinct mesh types were assessed. Because each type of mesh created a specific shape. To select the best type of mesh, the relevant parameters were first investigated. The mesh size was examined repeatedly. Because it offered exceptional mesh quality and simulated the model in a respectable length of time, the mesh size of 0.001m was selected. The kind of mesh was chosen based on the quality mesh that had been offered.

Water served as the primary phase and oil served as the secondary phase in a two-phase flow. The volume of fluid (VOF) was the chosen physical model. The VOF analysed the transit of all flow phases using two special equations. The turbulence model for the first test was chosen at this point in the procedure. Water and crude oil were sat in the flow's substance. The oil had 818 kg/m2 density and a viscosity of 1.75 kg/m2. The final step was to specify 0.005, 5000, and 50 as the time step, time, and iteration sizes for the simulation results. Both results were compared at the verification stage, along with the occupational area of the oil and the flow. The overall project was shown in Figure 8.

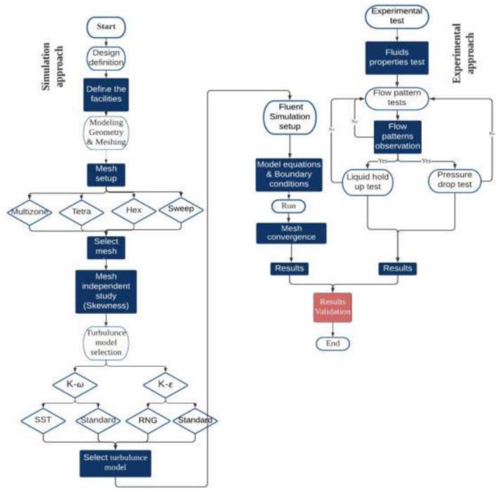


Figure 8: Overall flowchart of the project

Each mesh's size was measured by its face size; if it is reduced, the number of components would increase, and the model ran slowly. The element size must strike a compromise between precision and simulation duration. By improving the accuracy of the results by merely 1%, the simulation time could be increased by twofold or more. The element size was chosen to be 0.001m. The size was found to produce a good accurate result in a decent period and is a provably optimal size. Before the perfect size was discovered, numerous tests were conducted. The test results were shown in Figure 9, Figure 10, Figure 11, and Figure 12 but it was discovered that skewness scores increased with mesh size, indicating lower mesh quality. On the other hand, adopting a smaller size was not an option due to computational and temporal constraints.

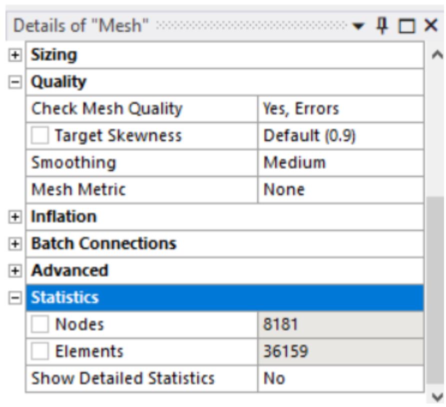


Figure 9: Mesh Independent Study

Outline

Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	Default (6.5648e-002 m)
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	Default (0.9)
Smoothing	High
Mesh Metric	None
Inflation	
Batch Connections	
Advanced	
Statistics	
Nodes	10257
Elements	9036
Show Detailed Statistics	No

Figure 10: Mesh Geometry (1e-003m)

Outline

Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	Default (6.5648e-002 m)
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	Default (0.9)
Smoothing	High
Mesh Metric	None
Inflation	
Batch Connections	
Advanced	
Statistics	
Nodes	1680008
Elements	1717232
Show Detailed Statistics	No

Figure 11: Mesh Geometry (2e-003m)

Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	Default (6.5648e-002 m)
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	Default (0.9)
Smoothing	High
Mesh Metric	None
Inflation	
Batch Connections	
Advanced	
Statistics	
Nodes	103399
Elements	95388
Show Detailed Statistics	No

Figure 12: Mesh Geometry (5e-003m)

Figure 13 showed the effect of the oil volume % on the flow adaptation region. Each area needed to be specified because it directly affected how the fluids flow. The primary discovery was that the volume portion of the oil decreased as air velocity rose, changing the pattern of the oil-water flow. The simulation and comparison included experimental data.

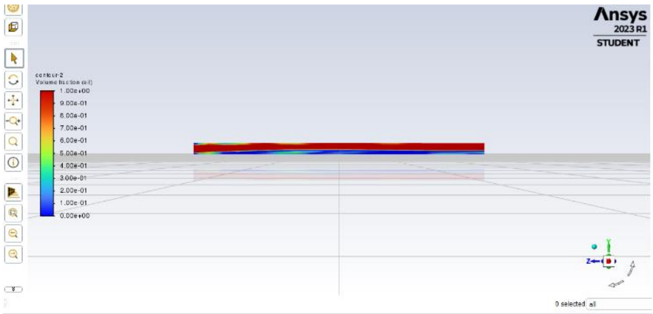


Figure 13: Region Adaption of 2 Phase

Inflation was a significant factor that was used to track the effects at the border. More cells could be accommodated around the edge of the wall using a technique called inflation, giving a more precise reading of the flow behavior. However, inflation had an impact on the meshing's number count, which speeds up calculation. It was found that there were five layers of inflation on the inlet and outflow. Mesh inflation was shown in Figure 14.

Details of "Mesh"

Export Preview Surface Mesh	No
Sizing	
Quality	
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No
Batch Connections	
Advanced	
Statistics	

Figure 14: Mesh Inflation

The experimental data were used to reproduce the flow pattern, estimated the pressure drop, and quantify the liquid holdup for the horizontal pipe. The model was initially built and created to pinpoint the critical elements and investigate how they influenced the results. The trial pipe was used as a model for creating the pipe and determining its characteristics. The mesh was a method to discretize the pipe into tiny cells for the simulated study of each cell. At this point, the mesh quality reflected the range of precision that each type of mesh offered. It was found that changing the quantity of meshes and the skewness value directly affected the mesh's quality. Different mesh sizes were attempted to find the model's best solution while taking simulation time into account. The fluid volume was calculated using the continuity equation as a function of the volume fraction of the phase, and an additional equation was incorporated to take the interface between the phases into consideration. The first model, which used the standard K-epsilon method, was used for the initial investigation of the correctness of the results.

The results of the tests that were run to accomplish the project's objectives were presented here. The results of these tests were compared with those of the experiments to ensure the validity of each test. The tests were well described using a variety of techniques. These protocols covered each test's experimental design, data collection methods, and data analysis. The tests listed below were run:

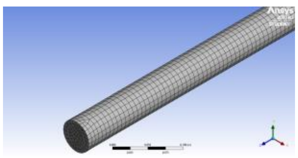
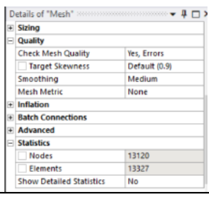
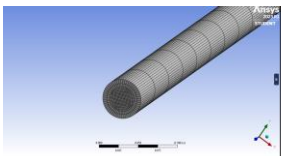
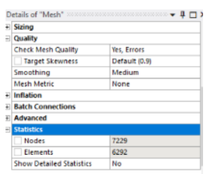
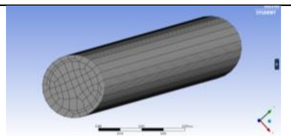
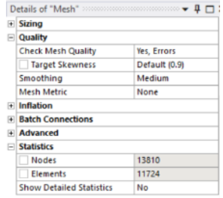
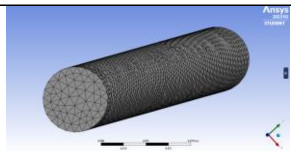
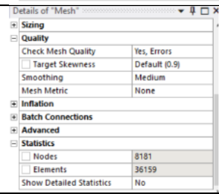
1. Selection of the optimum type of mesh for the pipe geometry.
2. Selection of turbulence type for the Malaysian waxy crude oil-water.
3. Evaluation of the pressure drop in a horizontal pipe.
4. Evaluation of the liquid holdup in a horizontal pipe.

This test was run to identify the ideal mesh type for the desired geometry. The mesh's quality was a crucial factor that affects the items' quality. More mesh components resulted in a more precise result. However, processing a larger mesh took more time on the computer. The quality of the mesh could not be assessed in many ways. However, the true process was to compare the number of mesh components to the skewness value, as demonstrated before.

To conduct the mesh independence study, experimental data were employed in the simulation to anticipate the flow pattern like those on the experiment. The mesh was made using the specs of the experimental pipe. To further improve the mesh's quality, simulation elements like mesh size, smoothing, and transition were applied. The pipe parameters in the lab setup were length, ID, roughness, material, and orientation.

A crucial element that directly influenced the simulation's outcomes was the mesh's quality. As a result, the testing of the various mesh types was done, and the results were displayed in Table 2. It was determined that the sweep produces a mesh with an ideal number of 6292 mesh components. Sweep provided the best option for greater accuracy because it provided the lowest skewness value. Furthermore. Tetrahedrons produce 36156 elements which may lead to less precise results.

Table 2: Optimum type of mesh selection

Method	Mesh Geometry	Mesh and Skewness Number
Hex Dominant		
Sweep		
Multizone		
Tetra		

Several factors and attributes affect the flow patterns of water-waxy crude oil. In a horizontal straight pipe, the flow pattern changed as the phase velocities change. CFD Fluent was used to model the discoveries of crude oil that is like air wax. The outcomes were validated by comparing the experimental flow patterns to the simulated flow patterns. The coordinate system that marked the boundaries of each type of flow with respect to the variation in the phase velocity was examined through several studies.

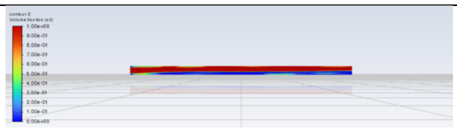
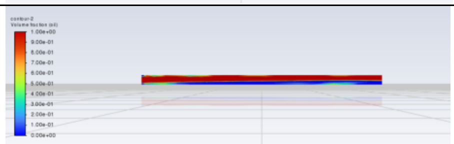
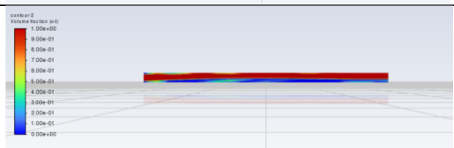
Results from the K-omega model were shown in Figure 15 to be more accurate than those from the other models. Therefore, K-omega was chosen to stand in for the numerous conceivable flow pattern types. The fluids' geographical adaptability was determined using the mixing velocity and pipe inner diameter. The software was set up at the inlet to assume that each phase occupies a certain amount of pipe. From the experiment, the water and oil's physical characteristics were taken.

The bubbly flow was tested. Due to the disparity in densities between the two phases, the simulation results displayed the flow in a side view with oil flowing at the bottom and air at the top. The oil might be seen to take up most of the pipe's cross-sectional area because the air only occupied a minor portion of the pipe. Water and oil were moving at velocities of 0.05 and 0.6 m/s, respectively. Due to the difference in densities between the two phases, the simulation results depicted a side view of the flow where oil was flowing on the bottom and air was flowing on top. Since the air only filled a small portion of the pipe, the oil could be seen to take up most of the cross-sectional space.

A quick action camera was used to find the flow patterns. The two flow patterns observed throughout the experiment were the scattered bubble flow pattern (DB) and the bubbly flow pattern. The bubbly flow pattern first surfaced in the experiment. This test's goal was to determine which turbulence model was the best. Additionally, the model that most closely resembles the flow in the experiment is chosen for the turbulence simulation. In this test, three turbulence models were employed since they were widely utilized in multiphase flow transportation equations.

Figure 15: K-Omega

Table 3: Turbulence Model Selection

Results	Flow Pattern
K-Omega (SST)	
K-Omega (Standard)	
RNG	

The frictional force, which was brought on by the pipe's flow resistance, resulted in a pressure drop in horizontal pipes. The pressure drop was put to the test to foretell how the air-oil flow would behave when it was affected by the pressure drop along the pipe. To observe the link between the pressure, drop and the air superficial velocity, tests were carried out using a constant oil superficial velocity and varied air superficial velocities. For conducting the pressure drop tests, the obtained models are recalled. All testing, however, exclusively utilized the k-omega turbulence model because it was chosen as the most effective model for two-phase flow.

Pressure reduction started to gradually increase when the flow changed from stratified to dispersed flow (intermittent area), specifically from STW to STSD&O with flow rate 46.08 m3/s and water fraction Cw of 0.8. In the oil-dominated area known as semi-dispersed flow with SDSE&TO, there was a modest increase in pressure loss when the flow changed from stratified to dispersed. This was because the clear pipe's interior surface had developed a thin layer of waxy crude oil under this state. The change of pressure drop with flow rate was shown in Figure 16.

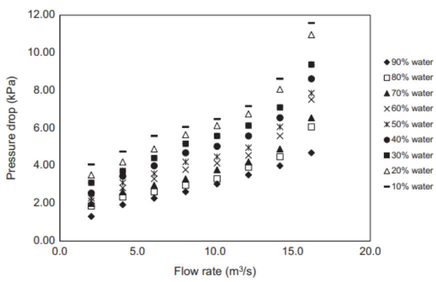


Figure 16: Pressure Drop vs Flow Rate

Liquid holdup is the in-situ volume fraction of water over all mixed liquids in a particular length of a test section. The LH discovered during the research. Water holdup, which depended on factors including liquid characteristics, flow pattern, pipe diameter, and pipe inclination, could typically not be estimated analytically but can be found experimentally. To catch the water holdup, two quick closing valves were fitted at either end of the transparent part. Carefully selecting the sort of quick closing valve to be utilized was necessary for accurate measurement of liquid holdup. This was the element that many researchers failed to consider while determining liquid holdup. The plots were shown in Figure 17 and Figure 18.

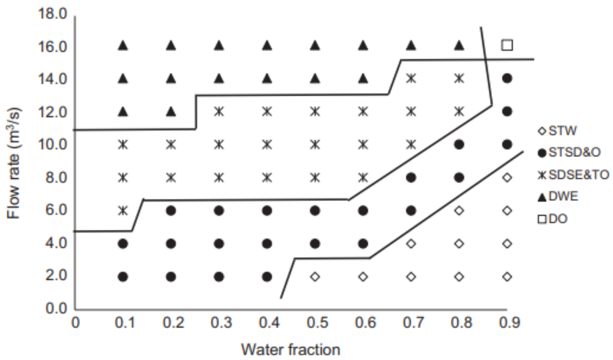


Figure 17: Waxy crude oil-water system at 30ºC

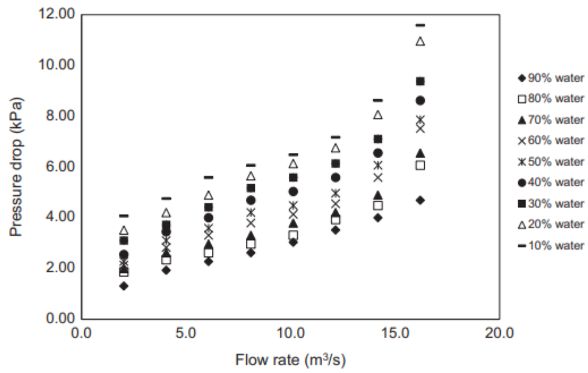


Figure 18: Pressure Drop vs. Flow Rate

ACKNOWLEDGMENT

We would like to thank Asia Pacific University for giving us the opportunity to conduct this paper.

REFERENCES

Al-Waily, M., Al-Baghdadi, M., & Resan, K. K. (2017). CFD Investigation of the Erosion Severity in 3D Flow Elbow During Crude Oil Contaminated Sand Transportation. Engineering & Technology Journal.

Awad, M. (2012). Two-Phase Flow. Intech.

Elseth. (2001). An Experimental Study of Oil/Water Flow in Horizontal Pipes. NTNU Open.

Faghri, A., & Zhang, Y. (2006). Transport Phenomena in Multiphase Systems. Elsevier.

Ghosh, S., & N. K. (1991). An arbitrary Lagrangian-Eulerian finite element method for large deformation analysis of elastic-viscoplastic solids. Computer Methods in Applied Mechanics and Engineering.

Hernández, S., & Vorobieff, P. (2020). Multiphase Flow: Computational & Experimental Methods. WII PRESS.

Huang, H., Sun, T., Zhang, G., Li, D., & Wei, H. (2019). Evaluation of a developed SST k-ω turbulence model for the prediction of turbulent slot jet impingement heat transfer. International Journal of Heat and Mass Transfer, 700-712.

Lee, J. W., Heo, H., Sohn, D. K., & Ko, H. S. (2021). Development of a numerical method for multiphase flows using an electrostatic model in a wire-mesh sensor. Engineering Applications of Computational Fluid Mechanics.

Lin, S., Kew, P., & Cornwell, K. (2001). Two-phase heat transfer to a refrigerant in a 1 mm diameter tube. International Journal of Refrigeration, 51-56.

Sanni, S. E., Olawale, A. S., & Adefila, S. S. (2015). Modeling of Sand and Crude Oil Flow in Horizontal Pipes during Crude Oil Transportation. Hindawi Journal of Engineering.

Tomar, G. (2015). Two-Phase Flow With Phase Change in Porous Channels. Journal of Thermal Science and Engineering Applications .