Research Article

Hyder M. Abdul Hussein*, Sabah T. Ahmed, and Laith Jaafar Habeeb

**Numerical and experimental investigation of two phase flow geometrical characteristics**

https://doi.org/10.1515/cls-2022-0018
Received Oct 30, 2021; accepted Feb 10, 2022

**Abstract:** This paper presents an experimental and numerical analysis of the effect of the geometric parameter on the two-phase flow (white kerosene-water) flow pattern system. The investigation was carried out using three lengths (1, 2 and 3) m of rectangular horizontal smooth channel and three channel heights of (5, 7.5 and 10) cm respectively. The flow conditions for the input water velocity (0.2 m/s) and the input kerosene velocity (0.1 m/s) for both measurements have been investigated. Two inlet techniques have been employed. Firstly, at the inlet, the kerosene was on top of the bath. Then, second, from the center, the kerosene inlets (water is above and below the kerosene). A numerical verification analysis was introduced using the ANSYS software using the method of volume of fluid (VOF) and mixture multiphase flow modeling coupled with the normal k-ε turbulence schemes. A collection of seven methods of CFD types is explored by running 224 instances. Comparisons were made between numerical and experimental works.

**Keywords:** Oil-water two phase flow, geometrical parameters, horizontal liquid-liquid flow pattern, CFD, type of inlet

**Nomenclature**

- \( C_{1\varepsilon}, C_{2\varepsilon} \) and \( C_{3\varepsilon} \) coefficient in standard k-ε model
- \( D_h \) hydraulic diameter [m]
- \( D_{o/w} \) Dispersion of oil in water
- \( D_{o/w&k/w} \) Dual dispersion of oil in water and water
- \( D_{o/w&k/w/o} \) Dual dispersion external body [N]
- \( g \) gravitational acceleration [m/s²]
- \( G_k \) production of turbulent kinetic energy [kg/ms³]
- \( G_b \) generation of turbulence kinetic energy due to buoyancy [kg/ms³]
- \( k \) Kinetic energy [m²/s²]
- \( p \) pressure [N/m²]
- \( S_k, S_\varepsilon \) Stratified by mixing at the interface flow
- \( ST & MI \) Time [s]
- \( X \) Mean velocity [m/s]
- \( \sigma_k, \sigma_\varepsilon \) Fluctuating dilatation in compressible turbulence to the overall dissipation rate [kg/ms³]
- \( \alpha \) Volume fraction
- \( \alpha_k \) Volume fraction of phase
- \( \varepsilon \) Turbulence dissipation [m²/s³]
- \( \mu \) Dynamic viscosity [kg/m·s]
- \( \rho \) Density [kg/m³]
- \( \rho_k \) Density of phase [kg/m³]
- *Corresponding Author: Hyder M. Abdul Hussein: Faculty of Engineering, University of Kufa, An Najaf, Iraq;
  Email: hyderm.alabady@uokufa.edu.iq
  Sabah T. Ahmed: Department of Aeronautical Engineering Techniques, Bilad Aliafidain University College, Diyala, Iraq;
  Email: dr.sabahtrik@bauc14.edu.iq
  Laith Jaafar Habeeb: Training and Workshop Center, University of Technology – Iraq, Baghdad, Iraq;
  Email: Laith.J.Habeeb@uotechnology.edu.iq

© 2022 H. M. Abdul Hussein et al., published by De Gruyter. This work is licensed under the Creative Commons Attribution 4.0 License
1 Introduction

In industry the multiphase flows extremely widely are used. Two or three phases were transportation as an immiscible fluid or may be in various physical flow conditions. These flows are occurring in much industrial sectors such as (petroleum, agricultural, food, pharmaceutical, etc.) [1]. Crude oil sources have outlying for the time being from most of strainer and processing centers. The high in pressure losses was major issue of dense and ordinary oil transmission in the horizontal of pipes line, which was high viscosity effect on friction, moreover, the acceleration or deceleration will cause increase these effect. The losses reduction was done by mix it with water. However, the multiphase oil-water pipe flow problem is very complex. It is highly dependent on many parameters such as input of the phases’ fraction, velocity of each phase, hydraulics diameter, and smoothness and ribbing inside the wall.

There are number of experimental and analytical studies on liquid-liquid flow in conduit. Russell et al. [2] carried out study the liquid-liquid flow where find three flow patterns by visual observations; stratified flow, bubble flow and mixed flow which were the first on this field. Charles et al. [3] investigated flow patterns, holdup ratios and pressure gradients, which had been the flow patterns Ao/w, So/w, Bo/w and Bw/o, and Do/w phase were observed. Karabelas [4] carried out a study to measure size spectra water dispersed in hydrocarbons with various flow rates. In parallel, Arirachakaran et al. [5] investigated pressure drop, flow rate, input water fraction, in-situ holdup, mixture temperature, and flow pattern, which were developed maps and two models to prediction pressure-gradient. Soleimani et al. [6] discussed observation of the flow pattern and pressure drop with two mixer elements were used. Trallero et al. [7], and Chakraborti et al. [8] introduced a two group of liquid-liquid flow patterns were called segregated and dispersed flow, which it were a different classification for flow patterns, and the segregated flow divided at stratified and intermittent flow. Fairuzov et al. [9] investigated by using a multi-point sampling probe a flow pattern transitions in horizontal pipelines carrying oil-water mixtures. Elseth [10] studied the behavior of the simultaneous focus have been on stratified and dispersed flow with pressure drops and local phase fractions of oil/water in horizontal conduits. Lovick and Angeli [11] investigation of the pressure drops, void fraction and flow pattern for two liquids. It was performed test to continuous of dual liquids in a horizontal pipe which was shown at middle mixture velocities. Bannwart et al. [12] studied the flow patterns for vertical and horizontal formed by heavy crude oil and water, which obtained for horizontal test section (stratified, bubbly-stratified, dispersed bubbles, and annular). Vielma et al. [13] collected several information were obtained on flow patterns, gradient of pressure, volume fraction and droplet size as function of flow patterns, which were used in flow description and validity of an oil-water model. Kumar et al. [14], Ismail et al. [15] and Zhai et al. [16] carried out experimental studies to obtain pressure gradient and flow structure for oil and water. Tan et al. [17] studied the effect of oil viscosity, pipe diameter, pipe material, and oil type on the flow pattern effect.

For the Computational fluid dynamics (CFD) part, Brauner and Maron [18] considered a parametric studied for wide ranges as density and viscosity relations to discuss the stability convergence to various extremes. Brauner [19] predicted the flow transition boundary of stratified to dispersed flow successfully. Gao et al. [20] utilized VOF with RNG $k$-$\varepsilon$ turbulence models combined with a near-wall low-Re to a stratified oil–water for steady-state flow were analyzed, which Pressure loss and hold-up were predictions, such showed acceptable agreement with experimental data. Desmala et al. [21] used VOF method with unsteady flow to predict the flow pattern (plug, slug, stratified wavy, stratified mixed and annular) maps and radial distribution of volume fraction. Shi et al. [22] employed method of the volume of fluid model with the SST $k$-$\omega$ scheme to compare the results with experimental of oil-water flow, which obtained a various flow pattern as core annular, plug of oil in water, oil bubbles in water and dispersed flow. Burlutskii [23] analyzed three dimensional CFD using $k$-$\varepsilon$ turbulence model with standard wall function, which imposed classical Lagrangian method to obtain dispersed flow. Baotong et al. [24] developed a new correlation for immiscible liquid utilized least square technique to compare experimental data with drift-flux correlation.

The objective of this paper is to introduce a new approach to investigate the development ability of oil-water flow in horizontal channel using geometrical parameters to predict the changes in flow pattern experimentally and numerically. A customized rig is built up for the experimental work. The numerical part is done utilizing CFD package FLUENT of ANSYS platform. Four influencing factors: effect of the length channel, the ratio of high to width, type of inlet flow, and the area of inlet ratio are chosen in this study to investigate flow patterns, pressure drops, and hold up.
2 Mathematical modeling and CFD

Volume of fluid (VOF) and mixture models are employed in this paper utilizing ANSYS FLUENT 19.0 over the other methods [25] due to its ability to predict the factors of each model in one hand. On the other hand, its simplicity and computational low cost. The CFD simulation carried out utilizing VOF and mixture models for different horizontal channels dimensions. Some assumptions are needed before proceeding to the modeling. These assumptions are introduced to simplify the proposed model solution.

2.1 Assumptions

The fluid flow is considered two dimensional, turbulent, and incompressible, no chemical reaction and the fluid properties are considered as constants.

2.2 The standard k-ε model

Based on model transport equations for the turbulence kinetic energy (k) and dissipation rate (ε), the prediction of this model shows good results according to [27]. For incompressible flow, the model has the following form, [21] and [26]:

\[
\frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho k \mathbf{u}) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + \varepsilon \tag{1}
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \nabla \cdot (\rho \varepsilon \mathbf{u}) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{k}{\varepsilon} \left( G_k + C_{3\varepsilon} G_{\varepsilon} \right) - C_{2\varepsilon} \rho \frac{\varepsilon^3}{k} + S_\varepsilon \tag{2}
\]

where \( \sigma_k \) and \( \sigma_\varepsilon \) are turbulent Prandtl numbers for k and \( \varepsilon \) respectively; \( Y_M \) represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate; and \( S_k \) and \( S_\varepsilon \) are the source terms \( G_k \) represents the generation of turbulent kinetic energy due to the mean velocity gradient.

2.3 VOF model theory

The capture of phase distributions was employed the volume of fluid (VOF) model carry out in the FLUENT. The phases have been engaged conservation equations with single set at the VOF model. Conservation equation was considered without mass transfer and phase change which should be an isothermal system. The equation of mass can be written as:

\[
\frac{\partial (\rho)}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{3}
\]

The momentum equation is as follows:

\[
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \left( \mu \left( \nabla \mathbf{u} + \nabla \mathbf{u}^T \right) \right) + \rho g + \mathbf{F} \tag{4}
\]

The parameters \( p, F, \text{and} \rho g \) interaction in dispersed flow will be denoted as Static pressure, external body force and gravitational body force, respectively. The primary phase (water) and the secondary phase (oil) was tracked interface together, which solution the equation of conservation for secondary phase (oil) the void fraction, \( \alpha_o \). The equation of the volume fraction is:

\[
\frac{\partial (\rho_o \alpha_o)}{\partial t} + \nabla \cdot (\rho_o \alpha_o \mathbf{u}) = 0 \tag{5}
\]

The volume fraction equation was not solved for the primary phase (water). The volume fraction of the primary phase was determined by the constraint:

\[
\alpha_o + \alpha_w = 1 \tag{6}
\]

Oil or water were filled the control volume completely, the volume fraction of the phase has a value of 0 or 1, so that the interface within control volume between 0 and 1.

The properties of matter in the transport equations were finding in each control volume by the presence of the component phases. For two phase flow (oil-water), that the density and viscosity in each cell was:

\[
\rho = \alpha_o \rho_o + \alpha_w \rho_w \tag{7}
\]

\[
\mu = \alpha_o \mu_o + \alpha_w \mu_w \tag{8}
\]

2.4 Mixture model theory

The mixture model can be used in different ways, which was a simplified multiphase model can be applied with different phase’s flows at different velocities. In addition, it is applicable to model homogeneous multiphase flow and to calculate non-Newtonian viscosity, which can model fluid or particulate for two or more phases when solved both the continuity equation and the momentum equation for the mixture. Furthermore, the mixture model can be a combination of dispersed phase and the continuous phase. The conservation equations used are the mass equation:

\[
\frac{\partial (\rho_m)}{\partial t} + \nabla \cdot (\rho_m \mathbf{u}_m) = 0 \tag{9}
\]
where $\bar{u}_m$: is the mass-averaged velocity and it is given by:

$$\bar{u}_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k u_k}{\rho_m}$$  \hspace{1cm} (10)

and $\rho_m$ is the mixture density calculated by:

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k$$  \hspace{1cm} (11)

Where $\alpha_k$ is the volume fraction of phase.

Moreover, the momentum equation:  

$$\frac{\partial}{\partial t} (\rho_m u_m) + \nabla \cdot (\rho_m u_m u_m) = -\nabla p$$  

$$+ \nabla \cdot \left[ \mu_m \left( \nabla u_m + \nabla u_m^T \right) \right]$$

$$+ \rho_m g + F$$

$$- \nabla \cdot \left( \sum_{k=1}^{n} \alpha_k \rho_k u_{dr,k} u_{dr,k} \right)$$

Note that, if the slip velocity is not solved the mixture model is reduced to a homogeneous multiphase model.

### 2.5 Computational geometry

As mentioned previously, four influencing factors are chosen to study the effect on flow pattern, pressure drops, and hold up. These factors are as following:

1. Effect of the channel length chosen to be (1, 2, 3) m.
2. The ratio of high to width is (1/2), selected for the channel heights is (5, 7.5, and 10) cm.
3. Type of inlet flow; halt to half (water in the bottom and upper oil). The other type was the oil inlet from the centre and the water is on the bottom and top of the oil, see Figure 1.
4. The area of inlet ratio is 1:1. i.e. (1 water to 1 oil).

### 2.6 Mesh generation

The geometry and mesh were developed in this study done by Ansys design modeler and imported into FLUENT for the simulations. Mesh forms an important part of numerical solution, and must accept confirmed criteria to ensure an accurate and valid solution. The discretization grid is structured and uniform Figure 2. Three types of mesh were chosen in this study coarse, middle, and fine mesh for accuracy of the solution. The sizes of these mesh types were listed in Tables 1 and 2.

### 2.7 Test of grid independence

The grid generation is a necessary task in any analysis. The condition of Reynolds number is an important to selection the mesh size, which can be divided into three types (fine, medium, and coarse). It was carried out the test to get the acceptable mesh cell size for special geometry. Three mesh faces are considered for 2-D which are listed pervious in the Tables 1 and 2. All three mesh faces are used to plot the (y+) for the channel estimated from Eqs. (13)–(15) and the values depicted in the Table 3. The mesh types were tested with an inlet velocities (Usw=0.2) m/s and (Uso=0.1 m/s) flow conditions of full flow domain of liquid used on the same x-y plot.

The standard wall functions are applied in Fluent program supported at the work of Launder and Spalding [26] since it is well-known and widely used in industry. They

---

**Table 1:** Mesh sizes, the oil inlet above the water

| Length (m) | A (cell)     | B (cell)     | C (cell)   |
|------------|--------------|--------------|------------|
| 1          | 190,000      | 240,000      | 480,000    |
| 2          | 240,000      | 550,000      | 720,000    |
| 3          | 480,000      | 800,000      | 1,040,000  |

**Table 2:** Mesh sizes, the oil inlet from the center

| Length (m) | A (cell)     | B (cell)     | C (cell)   |
|------------|--------------|--------------|------------|
| 1          | 190,000      | 240,000      | 520,000    |
| 2          | 288,000      | 550,000      | 780,000    |
| 3          | 480,000      | 720,000      | 1,080,000  |

---
Table 3: Estimation of $y^+$ and standard deviation for mesh types and channel lengths

| Type           | Length (m) | Coarse mesh cell | Wall   | $Y^+$ (mm) | $\sigma^*$ |
|----------------|------------|------------------|--------|------------|------------|
| Coarse mesh    | 1          | 190,000          | Lower  | 5.846      | 0.6244     |
|                |            |                  | Upper  | 1.028      | 0.272      |
|                | 2          | 240,000          | Lower  | 4.031      | 0.474      |
|                |            |                  | Upper  | 1.414      | 0.126      |
|                | 3          | 480,000          | Lower  | 4.482      | 0.513      |
|                |            |                  | Upper  | 1.447      | 0.142      |
| Middle mesh    | 1          | 240,000          | Lower  | 4.441      | 0.414      |
|                |            |                  | Upper  | 1.088      | 0.352      |
|                | 2          | 550,000          | Lower  | 3.656      | 0.474      |
|                |            |                  | Upper  | 1.038      | 0.153      |
|                | 3          | 800,000          | Lower  | 5.576      | 0.345      |
|                |            |                  | Upper  | 1.465      | 0.914      |
| Fine mesh      | 1          | 480,000          | Lower  | 6.316      | 1.204      |
|                |            |                  | Upper  | 1.228      | 0.672      |
|                | 2          | 720,000          | Lower  | 6.066      | 1.100      |
|                |            |                  | Upper  | 1.234      | 0.429      |
|                | 3          | 1,040,000        | Lower  | 6.298      | 1.249      |
|                |            |                  | Upper  | 1.288      | 0.552      |

$\sigma^*$: Standard deviation

are provided as a default option in ANSYS Fluent. The law-of-the-wall for mean velocity yields is:

$$u^* = \frac{1}{k} \ln(Ey^*)$$  (13)

Where the dimensionless velocity is:

$$u^* = \frac{U_p C_p^{1/4} k_p^{1/2}}{\frac{y_p}{\rho}}$$  (14)

and the dimensionless distance from the wall is:

$$y^* = \frac{C_p^{1/4} k_p^{1/2} y_p}{\mu}$$  (15)

Where:

- $k = 0.4187$ (von Karman constant), $E = 9.793$ (empirical constant)
- $U_p =$ fluid mean velocity near wall cell centroid
- $k_p =$ turbulence kinetic energy near wall cell centroid
- $y_p =$ distance from the centroid near wall cell centroid
- $C_p = 0.09$, $\mu =$ fluid dynamic viscosity

3 Simulation setup

Different setups and geometries were chose in this study for investigation. The types of scheme and high grid of resolution were computationally expensive. In addition, for the objective of discretization, and depend on requirement of solution accuracy were different scheme used as implicit or explicit scheme

3.1 Boundary conditions

The kerosene and water were split inlet into the zone, which was the computational domain full of water, then tracking the two liquids in the domain. The fluid properties listed in Table 4. The velocity input were listed in Table 5, where the boundary condition are selected from Lovick and Angeli [11].

Table 4: Fluid properties

| Material: kerosene | Property         | Value(s) |
|--------------------|------------------|----------|
|                    | Density          | 780 kg/m³ |
|                    | Cp (Specific Heat)| 2090 J/kg–k |
|                    | Thermal Conductivity | 0.149 w/m–k |
|                    | Viscosity        | 0.0024 kg/m–s |

| Material: water    | Property         | Value(s) |
|--------------------|------------------|----------|
|                    | Density          | 998.2 kg/m³ |
|                    | Cp (Specific Heat)| 4182 J/kg–k |
|                    | Thermal Conductivity | 0.6 w/m–k |
|                    | Viscosity        | 0.001003 kg/m–s |
Table 5: Boundary conditions

| Boundary conditions                      | Setup conditions inlet-1 (water) | Setup conditions inlet-2 (kerosene) |
|------------------------------------------|---------------------------------|------------------------------------|
| Velocity magnitude                       | 0.2 m/s                         | 0.1 m/s                            |

These conditions were summarized in Table 4, which was used in experimental work. Input condition employed for all optimization cases to prepare for a numerical solution that will compare results with experimental measurements. At the inlet, a velocity-inlet boundary type is used in which the water and kerosene velocities and the liquid volume fraction are specified.

### 3.2 Numerical investigation

The discretization of finite volume method was employed in governing transport equations. About continuity equation, and pressure-velocity coupling, which were imposed the algorithms of PRESTO! and PISO originally developed by Patankar [28] and Issa [29] were used. Table 6 shows the numerical relaxation factors and the discretization schemes.

Table 6: Summary of the numerical solution solver setting

| Variable                      | Relaxation factor |
|-------------------------------|-------------------|
| Pressure                      | 0.3               |
| Density                       | 1                 |
| Body Forces                   | 1                 |
| Momentum                      | 0.6               |
| Volume Fraction               | 0.4               |
| Turbulent Kinetic Energy      | 0.7               |
| Turbulent Dissipation Rate    | 0.7               |
| Turbulent Viscosity           | 1                 |
| Discrete Phase Sources        | 0.5               |
| Pressure-Velocity Coupling    | PISO              |

| Discretization Scheme         |
|-------------------------------|
| Variable                      |
| Pressure                      | PRESTO!            |
| Momentum                      | First Order Upwind |
| Volume Fraction               | First Order, QUICK, Modified HRIC, Compressive Upwind |
| Turbulent Kinetic Energy      | First Order Upwind |
| Turbulent Dissipation Rate    | First Order Upwind |

Selection volume fraction equation schemes is very important for solving case of multiphase flow problem, employed more than an option (First order, QUICK, Modified HRIC, and Compressive) then the choices are summarized in Table 6 with relaxation factors used for the numerical simulations. Numerical convergence was assumed when the residuals of continuity, momentum, turbulence and volume fraction equations were lowered by (10E–5) of magnitude. Turbulence intensity and hydraulic diameter were selected in the entrance, when turbulence model was active.

### 4 Experimental setup

An experimental rig is built specifically for the study. The cross-section area was circular for pumping liquid in the pipe line but the channel has been a rectangular cross section area. Furthermore, was specially designed, an adjustment to fit circular cross section to rectangular cross section as seen in Figure 3. Parts of the rig are shown in Figure 4. Rectangular channel was manufactured to observe the flow of liquid-liduids with test section length of (3.6 m). It was provided with two storages tanks (water 2000 L, and oil 250 L), and at the end connected with separated tank has a (500 L). Oil and water were used with physical properties listed in Table 4. It connects with oil and a water pumps which were transport the two fluids from the storage tanks to the rectangular channel.

**Figure 3: Adjustment for circular to rectangular cross section**
Characteristics of pumps selected for a water pump of (120 L/min) and for oil pump of (75 L/min). Moreover, the oil and water transmitted from the tank to the test section. Kerosene was injected from the center, and the water injected from the lower and the upper layers with inclination angle of 8° with respect to the horizontal axis. The two liquids were mixed at the beginning of the channel entrance.

Moreover, the volumetric flow rates for liquids were controls by using calibrated turbine flow meter (Sorekarain 1") and gear flow meter (Bre 400).

5 Results and discussion

Two types of experiments were assumed. The first will be called (inlet the kerosene above the water) and the second will be called (inter kerosene from the center). Three heights of channel were assumed (5, 7.5, 10) cm. First of all, \( (D_h = 3.33 \text{ cm}) \) with the water velocity is settled at \( (0.2 \text{ m/s}) \) and kerosene velocity at \( (0.1 \text{ m/s}) \) applied at all runs of more than 224 cases (see Table 7).

### Table 7: Cases numbers were solved

| Types                        | No. of cases |
|------------------------------|--------------|
| Inlet the kerosene above the water | 10 7.5 5 cm cm cm |
| 1 m                          | 21 - -       |
| 2 m                          | 21 21 -      |
| 3 m                          | 21 21 7     |
| Inlet the kerosene from the center | 10 7.5 5 cm cm cm |
| 1 m                          | 21 - -       |
| 2 m                          | 21 21 -      |
| 3 m                          | 21 21 7     |

[7] used \( (D_h = 5.013 \text{ cm}) \) to perform experimental flow pattern map, which appear at \( (U_{sw} = 0.2 \text{ and } U_{so} = 0.1 \text{ m/s}) \) stratified smooth flow pattern when compared with [8] who used \( (D_h = 2.54 \text{ cm}) \) shown stratified wavy at same velocities. Two models are applied, the mixture models contain four schemes and the VOF model used with three schemes listed in Table 8.

### Table 8: Schemes used in models

| Models          | Schemes                                      |
|-----------------|----------------------------------------------|
| Mixture model   | Mixture-implicit-no slip-first order         |
|                 | Mixture-implicit-no slip-quick               |
|                 | Mixture-implicit-slip-first order            |
|                 | Mixture-implicit-slip-quick                  |
| VOF model       | VOF-implicit-sharp-dispersed-compressive     |
|                 | VOF-implicit-sharp-compressive-modified HRIC |

5.1 One-meter channel

The tests made with one-meter length channel did not show any change in flow pattern with chosen other models or changes in the schemes. The analysis to this group was rapid and no disruption in the progress of the iteration.

5.2 Two-meters channel

Compared to the real behaviour in the previous studies, the best settings must be selected in the mathematical analysis. The results show good similarity in showing clearly visible separating layer from the analysis the split layer between the liquids (oil and water).

Reduction the height of channel led to a faster convergence of the solution encouraged the use the residual 10^{-4}
instead of $10^{-3}$. An analysis of the cases, shown in Table 7, was simpler and less complicated. The convergence was proceeding at a good pace. It requires some adjustments to volume fraction until reach the final convergence.

For the cases in Table 8 the issue more complicated and requires intervention to adjust the settings of pressure, momentum and volume fraction due to the residual was not progressing regularly.

Changing the accuracy of the solution from $(10^{-3}$ to $10^{-4})$ is the most recent change in the flow patterns. By increasing the steps of the solution, the breakup of the flow pattern changes. This indicates the instantaneous change of the phase, which is a result that matches the experimental fact corresponding to the computer analysis.

For cases of Table 9, the flow pattern shows a stratified for two liquids in the channel with some changes only in the entrance. Using $(L = 2 \text{ m}, H = 10 \text{ cm})$, the results does not show any change in the flow pattern during progression of the flow inside the channel.

Table 10, shows distinct results for the flow pattern of $(L = 2 \text{ m}, H = 7.5 \text{ cm})$. The results show the change in the flow pattern for the low mesh size channel which transfers from straight to random flow patterns which indicates a very important phenomena that, for some cases the separate layer shows mixing the two liquids together.

Table 11 shows the inlet of kerosene from the middle for $(L = 2 \text{ m}, H = 10 \text{ cm})$. The flow patterns show some differences through the channel.

**Table 9:** Inlet the kerosene above the water for two-phase flow patterns, $(L = 2 \text{ m}, H = 10 \text{ cm})$, mesh (720,000)

| Types of scheme                  | Flow pattern |
|----------------------------------|-------------|
| mixture-no slip-first order      |             |
| mixture-no slip-quick            |             |
| mixture-slip-first order         |             |
| mixture-slip-quick               |             |
| VOF-sharp dispersed-compressive  |             |
| VOF-sharp-compressive            |             |
| VOF-sharp-modified HRIC          |             |

**Table 10:** Inlet the kerosene above the water for two-phase flow pattern, $(L = 2 \text{ m}, H = 7.5 \text{ cm})$, mesh (780,000)

| Types of scheme                  | Flow pattern |
|----------------------------------|-------------|
| mixture-no slip-first order      |             |
| mixture-no slip-quick            |             |
| mixture-slip-first order         |             |
| mixture-slip-quick               |             |
| VOF-sharp dispersed-compressive  |             |
| VOF-sharp-compressive            |             |
| VOF-sharp-modified HRIC          |             |
Table 11: Inlet the kerosene from the middle for two-phase flow patterns, \((L = 2 \text{ m}, H = 10 \text{ cm})\), mesh (780,000)

| Types of scheme                        | Flow pattern |
|----------------------------------------|--------------|
| mixture-no slip-first order            |              |
| mixture-no slip-quick                  |              |
| mixture-slip-first order               |              |
| mixture-slip-quick                     |              |
| VOF-sharp dispersed-compressive        |              |
| VOF-sharp-compressive                  |              |
| VOF-sharp-modified HRIC                |              |

Table 12: Inlet the kerosene from the middle for two-phase flow patterns, \((L = 2 \text{ m}, H = 7.5 \text{ cm})\), mesh (780,000)

| Types of scheme                        | Flow pattern |
|----------------------------------------|--------------|
| mixture-no slip-first order            |              |
| mixture-no slip-quick                  |              |
| mixture-slip-first order               |              |
| mixture-slip-quick                     |              |
| VOF-sharp dispersed-compressive        |              |
| VOF-sharp-compressive                  |              |
| VOF-sharp-modified HRIC                |              |

The results in the Table 12 show that the flow was identical without any change in the flow pattern and certainly longer time needed to complete the solution.

5.3 Three-meters channel

The same inlet velocities were used in this analysis, so the effect of different densities was evident. Apparently, for the three-meter channel, when comparing the current results with previous results, there is a better behavior flow in this range of flow conditions.

The results in Table 13 were shows a little difference. It shows a repeated flow pattern with different models and scheme of analysis. Displays a pattern was similar to stratified flow pattern, and this was helped by increased length of the channel. Knowing that, by increasing the accuracy of mesh, there is no change in the pattern.

Table 14 of the channels \((L = 3 \text{ m}, H = 7.5 \text{ cm})\), presents new cases that have been worked to know the effect of reducing the height and increasing the length of the channel on changing the flow pattern. The results were change appeared in flow pattern in the first half of the channel, except the (VOF-implicit-sharp-modified HRIC) no change appeared.

The results in Table 15 show that the flow was identical without any change in the flow pattern even though the height was reduced.
Table 16 for kerosene inters from the middle for two phase flow patterns \((L = 3 \text{ m}, H = 10 \text{ cm})\) represent illogical behavior. The reason is that the difference between the two fluids necessitates the buoyancy of the oil over the water and not the other way around.

In Table 17, this setting (mixture-implicit-no slip-quick), and (mixture-implicit-slip-quick) show a behavior that approximates practical experiments, in other word, the mixture model detects the situation of the expected flow.

In Table 18, the patterns solved by (mixture-implicit-no slip-quick) and (VOF-implicit-sharp-modified HRIC) form are very similar. Same appearance for (mixture-implicit-no slip-quick) and (VOF-implicit-sharp-compressive) form. But there is a fundamental difference between the two forms. With deep inspection, the (mixture-implicit-no slip-quick) and (mixture-implicit-slip-quick) have mixing of layers whereas the (VOF-implicit-sharp-modified HRIC) and (VOF-implicit-sharp-compressive) form are not.

Table 19 shows a comparison between experimental and CFD work for (mixture-implicit-no slip-quick) scheme. The water velocity range from \((0.2-1.3) \text{ m/s}\) and for kerosene is from \((0.02-1.0) \text{ m/s}\). The four flow patterns (ST & MI, Do/w &w/o, Do/w, and Do/w&w) observed from experimental work. The results show a good agreement between the CFD and the experimental work.

### Table 13: Inlet the kerosene above the water for two-phase flow patterns, \((L = 3 \text{ m}, H = 10 \text{ cm})\), mesh \((1,040,000)\)

| Types of scheme                      | Flow pattern |
|--------------------------------------|--------------|
| mixture-no slip-first order          |              |
| mixture-no slip-quick                |              |
| mixture-slip-first order             |              |
| mixture-slip-quick                   |              |
| VOF-sharp dispersed-compressive      |              |
| VOF-sharp-compressive                |              |
| VOF-sharp-modified HRIC              |              |

### Table 14: Inlet the kerosene above the water for two-phase flow patterns, \((L = 3 \text{ m}, H = 7.5 \text{ cm})\), mesh \((1,040,000)\)

| Types of scheme                      | Flow pattern |
|--------------------------------------|--------------|
| mixture-no slip-first order          |              |
| mixture-no slip-quick                |              |
| mixture-slip-first order             |              |
| mixture-slip-quick                   |              |
| VOF-sharp dispersed-compressive      |              |
| VOF-sharp-compressive                |              |
| VOF-sharp-modified HRIC              |              |
Table 15: Inlet the kerosene above the water for two-phase flow patterns, (L = 3 m, H = 5 cm), mesh (1,040,000)

| Types of scheme          | Flow pattern |
|--------------------------|--------------|
| mixture-no slip-first order |             |
| mixture-no slip-quick     |              |
| mixture-slip-first order  |              |
| mixture-slip-quick        |              |
| VOF-sharp dispersed-compressive |         |
| VOF-sharp-compressive     |              |
| VOF-sharp-modified HRIC   |              |

Table 16: Inlet the kerosene from the middle two-phase flow patterns, (L = 3 m, H = 10 cm), mesh (1040000)

| Types of scheme          | Flow pattern |
|--------------------------|--------------|
| mixture-no slip-first order |             |
| mixture-no slip-quick     |              |
| mixture-slip-first order  |              |
| mixture-slip-quick        |              |
| VOF-sharp dispersed-compressive |         |
| VOF-sharp-compressive     |              |
| VOF-sharp-modified HRIC   |              |
### Table 17: Inter the kerosene from the middle for two-phase flow patterns, (L = 3 m, H = 7.5 cm), mesh (1,040,000)

| Types of scheme                        | Flow pattern |
|----------------------------------------|--------------|
| mixture-no slip-first order            |              |
| mixture-no slip-quick                  |              |
| mixture-slip-first order               |              |
| mixture-slip-quick                     |              |
| VOF-sharp dispersed-compressive        |              |
| VOF-sharp-compressive                  |              |
| VOF-sharp-modified HRIC                |              |

### Table 18: Inter the kerosene from the middle for two-phase flow pattern, (L = 3 m, H = 5 cm), mesh (1,040,000)

| Types of scheme                        | Flow pattern |
|----------------------------------------|--------------|
| mixture-no slip-first order            |              |
| mixture-no slip-quick                  |              |
| mixture-slip-first order               |              |
| mixture-slip-quick                     |              |
| VOF-sharp dispersed-compressive        |              |
| VOF-sharp-compressive                  |              |
| VOF-sharp-modified HRIC                |              |
Table 19: Inter the kerosene from the middle for two-phase flow pattern for (L = 3 m, H = 5 cm)

| Vso-Vsw | Pattern          |
|---------|------------------|
| 0.02-0.3 | ST & MI with wavy |
| 0.02-0.6 | Do/w & w/o       |
| 0.02-1.0 | Do/w & w/o       |
| 0.02-1.2 | Do/w            |
| 0.2-0.4  | ST & MI         |
| 0.2-1.0  | Do/w&w          |
| 0.6-0.2  | ST&MI           |
| 0.6-1.0  | Do/w& w/o       |
| 0.6-1.3  | Do/w & w/o      |
| 1.0-0.2  | Do/w& w/o       |
| 1.0-0.6  | ST&IM           |
| 1.0-1.3  | ST&IM           |
6 Conclusions

The main aim of this study is to investigate the influence of different parameters on the flow pattern of two-phase flow (liquid-liquid) in a rectangular channel. Experimental and numerical investigations were introduced in this work. A specialized rig is built for the experimental part while the numerical part achieved utilizing ANSYS Fluent package. A rectangular horizontal smooth channel of three lengths (1, 2 and 3) m and heights of (5, 7.5 and 10) cm respectively are employed. Kerosene and water are used to represent the fluid-fluid two phase flows. Two-inlet methods were employed. Firstly, the kerosene was interred on the top of water inlet. Then secondly, the kerosene inlets from the center (water is above and below the kerosene).

The Flow conditions have been investigated for input water velocity (0.2 m/s) and input kerosene velocity of (0.1 m/s) for all tests. The VOF models and mixture were used in the analysis. More than 226 different cases and scenarios are accomplished in this study. The results indicate a good agreement between the experimental and numerical work. As a future study, different inlet velocities for oil and water will be used for investigation. In addition, ribbed channel will be considered as a new investigation parameter.

Funding information: The authors state no funding involved.

Author contributions: All authors have accepted responsibility for the entire content of this manuscript and approved its submission.

Conflict of interest: The authors state no conflict of interest.

References

[1] Abdul Hussein HM, Ahmed ST, Habeeb LJ. Two-phase flow for gas-liquid, gas-solid, liquid-solid, and liquid-liquid in a horizontal smooth and turbulator conduit – A review. J Mech Eng Res Dev. 2020;43(7):26–50.
[2] Russell TW, Hodgson GW, Govier GW. Horizontal pipeline flow of mixtures of oil and water. Can J Chem Eng. 1959;37(1):9–17.
[3] Charles ME, Govier GW, Hodgson GW. The horizontal pipeline flow of equal density oil-water mixtures. Can J Chem Eng. 1961;39(2):27–36.
[4] Karabelas AJ. Droplet size spectra generated in turbulent pipe flow of dilute liquid/liquid dispersions. AIChE J. 1978;24(2):170–80.
[5] Arirachakaran S, Oglesby KD, Malinowsky MS, Shoham O, Brill JP. An analysis of oil/water flow phenomena in horizontal pipes. SPE Proceeding. Operating Symposium; 1989 Mar 13-14; Oklahoma City (OK), USA. 1989. p. 155-167.
[6] Soleimani A, Lawrence CJ, Hewitt GF. Effect of mixers on flow pattern and pressure drop in horizontal oil-water pipe flow. International Symposium on Multiphase Flow and Transport Phenomena; 2000 Nov 5-10; Antalya, Turkey. ICHMT, 2000. https://doi.org/10.1615/ICHMT.1997.IntSymLiqTwoPhaseFlowTranspPhen.60.
[7] Trallero JL, Intevep SA, Sarica C, Brill JP, Tulsa U. A Study of Oil / Water Flow Patterns in Horizontal Pipes. SPE Prod Facil. 1997;12(03):165–72.
[8] Chakraborti BD, Das PG, Ray S. Pressure Drop in Liquid-Liquid Two Phase Horizontal Flow. Chem Eng Technol. 2005;28(9):1005–9.
[9] Fairuzov YV, Arenas-Medina P, Verdejo-Fierro J, Gonzalez-Islas R. Flow pattern transitions in horizontal pipelines carrying oil-water mixtures: full-scale experiments. J Energy Resour Technol. 2000;122(4):169–76.
[10] Elseth G. An experimental study of oil/water flow in horizontal pipes [dissertation]. Porsgrunn: The Norwegian University of Science and Technology; 2001.
[11] Lovick J, Angeli P. Experimental studies on the dual continuous flow pattern in oil–water flows. Int J Multiph Flow. 2004;30(2):139–57.
[12] Bannwart AC, Rodriguez OM, Trevisan FE, Vieira FF, De Carvalho CH. Experimental investigation on liquid-liquid-gas flow: Flow patterns and pressure-gradient. J Petrol Sci Eng. 2009;65(1–2):1–13.
[13] Vielma M, Atmaca S, Sarica C, Zhang HQ. Characterization of Oil / Water Flows in Horizontal Pipes. SPE Annual Technical Conference; 2007 Nov 11-14; Anaheim, USA. 2000. p. 1-15.
[14] Kumara WA, Halvorsen BM, Melaen MC. Pressure drop, flow pattern and local water volume fraction measurements of oil–water flow in pipes. Meas Sci Technol. 2009;20(11):1–18.
[15] Ismail AS, Ismail I, Zoveidavianpoor M, Mohsin R, Piroozian A, Misnan MS, et al. Experimental investigation of oil–water two-phase flow in horizontal pipes: pressure losses, liquid holdup and flow patterns. J Petrol Sci Eng. 2015;127:409–20.
[16] Zhai LS, Jin ND, Zong YB, Hao QY, Gao ZK. Experimental flow pattern map, slippage and time–frequency representation of oil–water two-phase flow in horizontal small diameter pipes. Int J Multiph Flow. 2015;76:168–86.
[17] Tan J, Jing J, Hu H, You X. Experimental study of the factors affecting the flow pattern transition in horizontal oil–water flow. Exp Therm Fluid Sci. 2018;98:534–45.
[18] Brauner N, Maron DM. Stability analysis of stratified liquid-liquid flow. Int J Multiph Flow. 1992;18(1):103–21.
[19] Brauner N. The prediction of dispersed flows boundaries in liquid–liquid and gas–liquid systems. Int J Multiph Flow. 2001;27(5):885–910.
[20] Gao H, Gu H, Guo L. Numerical study of stratified oil – water two-phase turbulent flow in a horizontal tube. Int J Heat Mass Transf. 2003;46(4):749–54.
[21] Desmala AB, Vijayan V, Dasari A, Dasmahapatra AK, Mandal TK. Prediction of oil-water flow patterns, radial distribution of volume fraction, pressure and velocity during separated flows in horizontal pipe. J Hydrodynam. 2016;28(4):658–68.
[22] Shi J, Gourma M, Yeung H. CFD simulation of horizontal oil-water flow with matched density and medium viscosity ratio in different flow regimes. J Petrol Sci Eng. 2017;151:373–83.
[23] Burlutskii E. CFD study of oil-in-water two-phase flow in horizontal and vertical pipes. J Petrol Sci Eng. 2017;162:524–31.

[24] Baotong S, Rassame S, Nilsuwankosit S, Hibiki T. Drift-flux correlation of oil–water flow in horizontal channels. J Fluids Eng. 2018;141(3):31301.

[25] ANSYS, Inc. (2019). ANSYS fluent theory Guide 19.0.

[26] Launder BE, Spalding DB. The numerical computation of turbulent flows. Comput Methods Appl Mech Eng. 1974;3:269–89.

[27] Babakhani Dehkordi P. Experimental and numerical analysis of multiphase flow within horizontal pipeline with variable cross-sectional area [dissertation]. Milan: Politecnico di Milano; 2017.

[28] Patankar SV. Numerical heat transfer and fluid flow. New York: McGraw-Hill; 1980.

[29] Issa RI. Solution of the implicitly discretised fluid flow equations by operator-splitting. J Comput Phys. 1986;62(1):40–65.