25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

# COB-2019-0287
# NUMERICAL SIMULATION OF TWO-PHASE OIL-WATER FLOW IN HORIZONTAL PIPELINES: A REVIEW

**Ronaldo Luís Höhn**
School of Mechanical Engineering, University of Campinas
ronaldohohn@gmail.com

**Marcelo Souza de Castro**
School of Mechanical Engineering, University of Campinas
mcastro@fem.unicamp.br

***Abstract.****Liquid-liquid two-phase flows are present in numerous processes in nature and industry. Such flows have different flow patterns with different characteristics and the differences between the flow patterns have been studied experimentally and analytically by several authors. In addition to this, research with numerical simulations has been carried out, enabling the solution of the general governing equations which are of great complexity for these flows. However, some different results and predictions are reported by the researchers, who were divergent in numerous hypotheses during the description of liquid-liquid flows in their simulations. Thus, this work consists of performing a literature review on the current state of liquid-liquid two-phase flows simulation studies, mainly at horizontal pipelines, highlighting important concepts for studies of such flows using numerical techniques. In addition to this, a numerical simulations study is carried out in order to obtain the best combination and hypotheses for simulation of two-phase oil and water flows in horizontal pipelines.*

***Keywords:****liquid-liquid flow, flow patterns, two-phase flow models, Numerical simulation, M-CFD*

## 1. INTRODUCTION

Simultaneous flows of two immiscible phases are known as two-phase flows and are involved in various industrial and natural processes. It is possible to classify them in relation to the phases that compose the system, such as gas-liquid flows, solid-liquid, and liquid-liquid flows. Some common place where the gas-liquid flow is present are in bubble columns, cooling towers, oil-gas or water-gas flows in pipelines, industrial cooling and ventilation, production and transportation of natural gas and oil. The gas-solid or solid-liquid flows can be found in pneumatic transport, combustion and gasification processes, catalytic cracking, particulate matter transport, bioreactors and bubble-flushed beds. Finally, liquid-liquid flows are generally found in the transport of emulsions in the food industry and in the oil industry during oil transport and production.

When two fluids flow simultaneously in a pipe, they can assume different spatial configurations, also called flow patterns. This characteristics is one of the most remarkable of the two-phase flows, (Taitel and Dukler, 1976; Wang *et al.*, 2017). However, the liquid-liquid flow patterns are until now of great controversy, and further studies are still needed over the flow patterns and their characteristics. According to Wang *et al.* (2017), one of the main reasons that difficult the creation of a general flow pattern identification are because there is a great variability of viscosity ratios that are possible in this type of flow, and with that, different flow patterns classification are proposed by the authors, (Torres-Monzón, 2006).

Because this many possibilities and configurations of liquid-liquid flows, more studies have been carried out to better understand its characteristics. Empirical studies have been developed in search of to build a database, next to this, new phenomenological models have been described to determine and predict the characteristics of liquid-liquid flows such as pressure gradient and holdup (Volumetric fraction) (Wang *et al.*, 2017). Moreover, with increasing processing capabilities of computers, numerical studies have also been carried out. The simulation of these flows showed to be able to validate models, to expand the databases obtained, to obtain previous data for experiments, and also to predict whole two-phase flows characteristics.

With the increase of the application of CFD in liquid-liquid flows, different configurations and considerations are used by the authors for similar flows. In general, the results found with the numerical simulations are in agreement with the expected empirical data and flow patterns, which encourages the use of CFD for such cases. Several excellent papers in which commercial CFD software are used have been published, such as Ávila (2016), Desamala *et al.* (2014), Rodriguez and Baldani (2012), and another using opensource software alike Rzehak and Kriebitzsch (2015). However, for similar cases, different mesh configurations, turbulence models, and hypotheses were taken. So, in some cases, divergences between pressure gradient results, wave parameters and other desired predictions in relation to the empirical databases were observed. Based on this, this work revises some techniques used in numerical simulations of two-phase liquid-liquid flows, focusing on horizontal oil-water pipe flows and presents the current study of two-phase liquid-liquid stratified flow simulations, where simulations and experimental data on smooth stratified (SS), wavy stratified (SW) and stratified with mixture at the interface (ST&MI) will be performed.

## 2. Computational Fluid Dynamics in Liquid-Liquid Flows

To perform the simulation of any flow, a discretization of the conservation equations and the flow domain must be performed, making it possible to solve the problems numerically. Among the most used methods to perform these discretizations, the finite volume method is the most common method used between commercial and open source codes. This model is used because of its local conservation principle, dividing the region of the flow into small volumes (cells) and the equations that govern the flow are reformulated for each computational cell, forming a set of algebraic equations that are usually solved in interactive numerical methods (Andersson *et al.*, 2012).

It is common to treat the liquid-liquid two-phase flows of water and oil in horizontal and slightly inclined pipes as isothermal (Ávila, 2016). As a result, two conservation equations are important, the mass conservation equation and the equation of momentum. The mass conservation equation for two-phase flows for a phase $k$ can be given by Eq. (1),

$$\frac{\partial}{\partial t}\left(\varepsilon_k \rho_k\right) + \nabla \cdot \left(\varepsilon_k \rho_k \vec{V}\right) = \sum m_k \tag{1}$$

where $\varepsilon_k$ represents the volumetric fraction of each phase, $\rho_k$ the density, $V$ the superficial velocity of the phase and the term $\sum m_k$ characterizes the sum of mass transfers between phases, being equal to zero in situations without mass transfer.

The two-phase momentum equation can be defined such as, Eq. (2):

$$\frac{\partial}{\partial t}\left(\epsilon_k \rho_k \vec{V}_k\right) + \nabla \cdot \left(\epsilon_k \rho_k \vec{V}_k \otimes \vec{V}_k\right) = -\epsilon_k \nabla p + \epsilon_k \nabla \cdot \tau_k + \epsilon_k \rho_k \vec{g}_k + \sum_{j=1}^{n}\left(\vec{R}_{jk} + \dot{m}_{jk}\vec{V}_{jk}\right) + \varepsilon_k \rho_k \left(\vec{F}_k^{inter}\right) \tag{2}$$

where $\dot{m}_{jk}$ is the mass transfer between the $n$ phases of $j$ and the $k$ phase, $\vec{V}_{jk}$ represents the interfacial velocity between each phase $j$ with phase $k$. The force $\vec{F}_k^{inter}$ consists of the sum of the interfacial forces in several different contributions, such as lift forces, virtual mass forces and external field forces. The $\tau_k$ is the tensor of the shear forces for each phase and the term $\vec{R}_{jk}$ is the interaction term between the $j$ phases with phase $k$.

### 2.1 Models for Multiphase Flows

Different models are used to describe and calculate the physical charcateristics of multiphase flows in CFD. The choice of one model should take into account factors such as available processing capacity, characteristics of the cases to be studied, the numerical method used, agreement with other physical modeling models used together in the simulation, among other contour and control characteristics applied in the construction of the simulation.

The Eulerian-Eulerian model is the most widely used multiphase flow model, two models belonging to this approach are commonly used, the Euler-Euler model and the Volume of Fluid (VoF). The Euler-Euler approach structure is suitable for situations where the phases are mixed and the velocity of each phase is relevant. This model is based on the assumption that all phases are considered continuous and interpenetrating and all phases have to share the same space in the domain. The volumetric fractions of each phase may vary with time and space, however, since the volume of one phase cannot occupy or be occupied by the other, the sum of the volumetric fractions in time and space must always be equal to 1, (Paladino, 2005).

The phases are treated separately and sets of governing equations are solved for each phase. The interfacial surface has pressure fields and coefficients of interfacial transfer, these variables must be modeled. The resolution of this model requires the transport equations for the volumetric fraction of each phase to be solved numerically, (dos Santos, 2009; Ávila, 2016). The equation of mass conservation Eq. (3), the equation of the momentum Eq. (4) and the equation of the volumetric fraction Eq. (5) for this approach, for a phase $k$ are:

$$\frac{\partial}{\partial t}\left(\varepsilon_k \rho_k\right) + \nabla \cdot \left(\varepsilon_k \rho_k \vec{V}_k\right) = 0 \tag{3}$$

$$\frac{\partial}{\partial t}\left(\varepsilon_k \rho_k \vec{V}_k\right) + \nabla \cdot \left(\varepsilon_k \rho_k \vec{V}_k \otimes \vec{V}_k\right) = -\varepsilon_k \nabla p + \varepsilon_k \nabla \cdot \tau_k + \varepsilon_k \rho_k \vec{g}_k + S_k \tag{4}$$

$$\frac{\partial \varepsilon_k}{\partial t} + \nabla \cdot \left(\varepsilon_k \vec{V}_k\right) = 0 \tag{5}$$

The interfacial area $A_{\alpha\beta}$ between the phases is calculated by Eq. (6):

$$A_{\alpha\beta} = \frac{\varepsilon_\alpha \varepsilon_\beta}{d_{\alpha\beta}} \tag{6}$$

where, $d_{\alpha\beta}$ is the measure of a specified interfacial length scale and $\varepsilon_\alpha$ e $\varepsilon_\beta$ are the volumetric fractions of the phases $\alpha$ and $\beta$, respectively.

The other common approach used is the volume of fluid model (VoF). This is used for two or more immiscible fluids where the position of the interface is of great interest. The VoF model differs from the Euler-Euler approach since it solves a single set of transport equations using mixing properties, with no slip velocity. In this approach, an advection equation for the phase indicator function must be calculated numerically to control the interface. This characteristic causes some limitation when the method is used for systems with significant variations between phase velocities.The volume of fluid model was developed for flows with well-defined interfaces (Gopala and van Wachem, 2008), being an approach suitable for stratified flows, elongated bubbles and other flows with well defined surfaces.

The indicator function to locate the interface between phases acts similarly to the volumetric fraction. If a cell has only a single phase the value 0 or 1 is assigned. For a cell where the identifier function has a value between 0 and 1, the surface interface will be located in this cell and its mixing properties are calculated based on the value of the indicator function, (Ávila, 2016). The mass conservation Eq. (7), momentum equation Eq. (8) and the equation of phase indication Eq. (9) for the VOF approach are:

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \overrightarrow{V}) = 0 \tag{7}$$

$$\frac{\partial}{\partial t}(\rho_m \overrightarrow{V}) + \nabla \cdot (\rho_m \overrightarrow{V}\overrightarrow{V}) = -\nabla p + \nabla \cdot \tau + \rho_m \overrightarrow{g} + S_k \tag{8}$$

$$\frac{\partial \varepsilon}{\partial t} + \nabla \cdot (\varepsilon \overrightarrow{V}) = 0 \tag{9}$$

where the density of the mixture is calculated for each control volume by $\rho_m = \Sigma \varepsilon_k \rho_k$. Depending on the control volume, the volumetric fraction $\varepsilon$ and the velocity $\overrightarrow{V}$ may represent the mixture or a single existing phase.

### 2.2 Turbulence Models

In most of the oil-water two-phase flows at least one of the phases will present turbulence, a factor that must be considered since certain levels of turbulence can significantly influence the values of pressure gradient. There are three strategies for modeling turbulence in CFD, and these can be divided into two groups: simulations and modeling group. According to Zikanov (2010), in the simulations group, calculations are applied for the actual realizations of the flow, and in the modeling group, systems of equations are applied to solve the average flow quantities, without having to calculate the actual realization of the flow. The direct numerical simulation (DNS) and large eddy simulation (LES) methods are in the group of simulations, and the Reynolds averaged Navier-Stokes (RANS) are in the modeling group.

According to Ribeiro (2012), the turbulence models of two equations are the most used in the RANS approach because they present good convergence and reasonable responses in relation to the global flow behavior, requiring low computational capacity in relation to the other turbulence methods and models. The two-equation models that stand out are $k$-$\varepsilon$, $k$-$\omega$ and SST. The $k$-$\varepsilon$ model stands out in its practicality. However, the $k$-$\omega$ model is characterized as a good model for confined regions and has its deficiency far from the wall. The SST model, as suggested by Rzehak and Kriebitzsch (2015), appears as a combination of the $k$-$\varepsilon$ and $k$-$\omega$ models with characteristics varying according to the domain and flow characteristics.

Several studies with simulations of two-phase flows were performed with the $k$-$\varepsilon$ turbulence model and presented satisfactory results, such as the studies carried out by Ghosh *et al.* (2010), Rodriguez and Baldani (2012), Alagbe (2012), Desamala *et al.* (2014), Ávila (2016). According to Ávila (2016), this model is quite robust in relation to turbulence calculations, since they have a considerable number of adjustable constants. However, according to Paladino (2005), there are no well-functioning turbulence models that have been widely applied, as in single-phase flows. What happens is that the models applied to multiphase flows are extensions of models used in single phase flows, in which some phenomenological differences are employed.

### 2.3 Computational Meshes

To perform the numerical simulations, the equations governing the flow and the domain must be discretized. The discretization of the flow domain is obtained by dividing the flow region into a number of control volumes, also known as cells or elements, forming a computational mesh.

Three types of meshes that are commonly used in the CFD, the structured mesh, the unstructured mesh, and the hybrid mesh. A structured mesh is restricted by containing quadrilateral cell in 2D or hexahedra cells in 3D. Thus, they are characterized by regular connectivity that can be expressed by two- or three-dimensional, which saves space. The unstructured mesh is characterized by irregular connectivity between the elements, allowing any format cells to be used. This makes easy the representation and refinement of the domain. However, the required computational capacity tends to increase considerably with unstructured meshes (Maliska, 2004).

Recently, multi-block techniques are being used in CFD simulations, this method consists in dividing the region of the flow into blocks and zones. Each block is meshed with different characteristics to better represent the region of the flow. This zoning allows structured meshes to be applied with different characteristics in the same domain, which tends to facilitate the interactions and decrease the computational cost in relation to the unstructured mesh (Ali *et al.*, 2017).

Many of the works with numerical simulation of liquid-liquid two-phase flows used unstructured meshes and presented satisfactory results. According to Ávila (2016), the reason for using this type of mesh is because it is recommended for non-regular geometries or those which need a greater refinement in a region of interest. The creation of the entire pipe geometry or only of one section tends to vary with the flow pattern to be simulated. An exemple of this can be given by seeing the study of Ghosh *et al.* (2010), which simulated the annular flow pattern reproducing the entire section of the tube in its computational domain, differently Ávila (2016), in order to reduce the computational cost, the pipe was generated in half lengthwise to simulate the stratified wavy flow pattern.

### 2.4 Computational Fluids Dynamics in liquid-liquid flows

In this section some numerical simulation works of two-phase liquid-liquid flows are presented, as well as some important points mentioned by these studies.

Ghosh *et al.* (2010) using FLUENT commercial software 6.3.26 simulated vertical downward annular flow with a high viscosity oil core surrounded by water. The 3D model representing the pipeline had an internal diameter of 0.012 $m$ with a length of 0.48 $m$. Considering the oil as always laminar, due to high viscosity and considering the water as always turbulent, the authors applied individual inlets for each phase. To solve the turbulence present in the water phase, they used the $k - \varepsilon$ model. The authors state that this flow pattern presents well-defined interfaces, base on this they decide to use the Euler-Euler approach. Thereby, they observed a great variation in the radial velocity at the interface, which became more evident as the flow progressed. They also identified that the oil volumetric fraction remained constant after a length of 22 diameters. The results were compared with experiments and showed that CFD simulations can predict the hydrodynamic characteristics of annular flow adequately.

Rodriguez and Baldani (2012), with a focus on the wavy stratified flow pattern with no dispersion at the interface and in the turbulent-laminar regime, suggested a new closure relation for the interfacial friction factor based on the equivalent sand roughness concept. A new equation according to the number of Eötvös, holdup and contact angle was proposed for the transverse interface shape. The proposed phenomenological model was validated with its empirical predictions, current data, literature models and CFD results. The simulations were performed using commercial ANSYS CFX 10 software, where water and oil properties were defined as $\rho_w$ = 997 $kg/m^3$, $\rho_o$ = 824,3 $kg/m^3$, $\mu_w$ = 0,89 $mPa \cdot s$ and $\mu_o$ = 214,6 $mPa \cdot s$. The discrete pipeline domain had an internal diameter of 0.026 $m$ for a length of 1 $m$, with a glass surface and a hybrid mesh close to the wall. The Eulerian approach was chosen as a model for two fluids, applying the laminar model for the oil and for water the turbulence model $k - \varepsilon$. The calculated and empirically observed waves were also satisfactorily represented by the numerical simulation, which indicates good agreement between the results. It was also possible to observe the concave interface between the phases, a situation that was expected due to the simulated configurations. Finally, the qualitative results and the prediction of the holdup obtained by the numerical simulation were satisfactory. However, the results obtained in relation to the pressure gradient were inconsistent.

Alagbe (2012) conducted empirical and numerical studies on water-sand, oil-water and oil-water-sand flows in 5 $m$ long pipes with 0.0254 $m$ in diameter. Performing mesh sensitivity tests based on the pressure gradient of liquid-liquid flow, found as ideal a 3D mesh composed of 480000 cells, with hexahedral structure. With the focus on the validation of turbulence models for oil and water flows, the choice of turbulence model was performed comparing experimental data with pressure drop values found by four turbulence models. The authors found that the prediction of the standard $k$-omega and low-Reynolds-$k$-omega models was lower than the empirical data, and the Standard $k$-epsilon and low-Reynolds-$k$-epsilon models presented higher values of pressure drop than the experiments. Finally, the $k-\varepsilon$ models were used because of their greater applicability in the engineering, but some modifications in these models were made in order to improve the performance in the predictions of the flow characteristics. All flow patterns seen in their tests were also observed during CFD simulations. In general, the results found with the numerical simulations were according to the empirical data.

Studies with oil and water horizontal flow simulations with a viscosity ratio of 107 and a density ratio of 0.89, the wavy stratified, mixed stratified and annular flow patterns were successfully predicted using commercial software FLUENT 6.2 by Desamala *et al.* (2014). The geometry of the tubing used in its studies had the internal diameter of 0.025 $m$. The two phases are injected by a $T$, and the computational domain was constructed with a two-dimensional mesh. Hydrodynamic characteristics were simulated and compared with empirical results showing in good agreement. However dispersed flow patterns were not predicted with CFD techniques.

With the focus on the wavy stratified flow pattern, Ávila (2016) used the Ansys-CFX 14.0 software to obtain data of oil volumetric fraction, pressure drop, amplitude and interfacial wavelength, which compared with results of the proposed phenomenological modeling in his study and with experimental databases of Pereira (2011) and Castro (2013). Considering the flow of oil always laminar and the flow of water as turbulent. The $k - \varepsilon$ model was applied in the simulations, using the backward-euler approach to describe multiphase physics, along with a drag coefficient of 0.44. The tubing represented in the numerical simulations had a diameter of 26 $mm$ and a length of 2 $m$, and the inlet region had the $Y$ shape, so that the phases had individual inlets. In order to reduce the computational cost, the pipe was generated in half lengthwise. The domain was discretized using unstructured meshes with the region of the center of the pipe being the most refined, seeking a greater precision in the interface region. Through a mesh sensitivity test, a mesh of 200000 nodes was established to obtain the volumetric fraction and the pressure gradient. However, it was observed that to evaluate the wave parameters it was necessary to use a much thinner mesh, choosing a transition process with an intermediate mesh, in order to decrease computational cost and maintain data reliability. The results of CFD in relation to the volumetric fraction showed good agreement with the phenomenological models and empirical data. However, the pressure gradient data did not present satisfactory results for horizontal cases, different from inclined cases that presented pressure gradient forecasts closer to the empirical results. The results in relation to the wave parameters show only qualitative agreement, failing to predict these parameters.

## 3. Numerical Simulations

Numerical simulations have been carried out in order to obtain the ideal settings for the resolution of CFD simulations for two-phase oil-water flows. The open source software OpenFOAM and the commercial software Ansys-Fluent are used in order to obtain non-software dependent results. For horizontal pipes of one, two and three inch initially three flow patterns are being investigated, the stratified smooth (SS), wavy stratified (SW) and stratified with interface mixing (ST & MI). The results of the two-phase flow simulations for one-inch pipes will be compared to the Castro (2013) data, the results of the two and three-inch simulations will be compared to work data that is still in progress by the same research group. In this article, the simulation of the smooth stratified case is described.

It was assumed as initial hypotheses for the numerical solution process: Newtonian fluids, the flows are incompressible and isothermal, transient regime, three-dimensional, with no mass transfer between the fluids, with an active gravitational force ($g = -9.81\ m/s^2$) and symmetrical with respect to the YZ plane. For the simulation of the smooth stratified flow (SS), the laminar flow hypothesis is assumed for both water and oil phases, as Castro (2013) suggests since this flow pattern occurs for low superficial water and oil velocities. However, for the simulation of stratified wavy (SW) and stratified with interface mixing (ST & MI) cases it will be considered the oil phase as laminar and the water phase as turbulent.

For both phases the pressure of 1 atm and temperature of 26$°C$ were assigned as reference. Following the experiments carried out by Castro (2013), the water properties employed were: density of $\rho_w = 988\ kg/m^3$, with a dynamic viscosity of $\mu_w = 0,00101\ kg/m \cdot s$, heat expansion of $0,000257\ K^{-1}$, specific heat capacity of $4181,7\ J/kg \cdot K$ at constant pressure and molar mass of $18,02\ kg/mol$. The properties of the oil assigned were: density of $\rho_o = 854\ kg/m^2$, dynamic viscosity of $\mu_o = 226$ centipoises, and molar mass of $1,0\ kg/kmol$. The interfacial tension for the simulations related to the experiments of Castro (2013) was $\sigma = 0,045\ N/m$.

### 3.1 Geometry and Mesh

The pipe geometry was generated with the format of "Y", with two inlets, the upper inlet for the oil and the lower for the water (Fig. 1), this configuration of two inlets in "Y" is done in some of the horizontal liquid-liquid flow experiments and was successfully used in similar studies of numerical simulations. This configuration allows control of the velocities/flows separately of both fluids and does not require a minimum length for the separation of water and oil for the formation of the segregate flow patterns desired. The pipe geometry has been generated sectioned longitudinally in order to minimize the computational cost since for the stratified flows the symmetry hypothesis in the flow can be assumed. The main pipe length is two meters for the three diameter variations (1, 2 and 3 in). The inlet pipes are inclined at 45° to the main pipe and have the same diameter.

With the proposals to use two different CFD software, it was defined to use the same computational mesh for both software when simulating the same data. Therefore, the snappyHexMesh tool of OpenFOAM is used to created different meshes and export them to Ansys-Fluent, facilitating the comparison of results to be obtained between the two software.

The snappyHexMesh tool is a hexadecimal and split-hexahedra automatic splitter, generating the final mesh from a base mesh already created with the blockMesh tool, adapting and refining it based on triangular geometric surfaces such as Stereolithography (STL) or Wavefront Object (OBJ) format. This tool allows flexible control of regions of refinements and meshes qualities, guaranteeing a final mesh quality to be executed in OpenFoam (Greenshields, 2018). Providing a refinement of the interface regions between water and oil, required mainly for the simulations of flows patterns wave

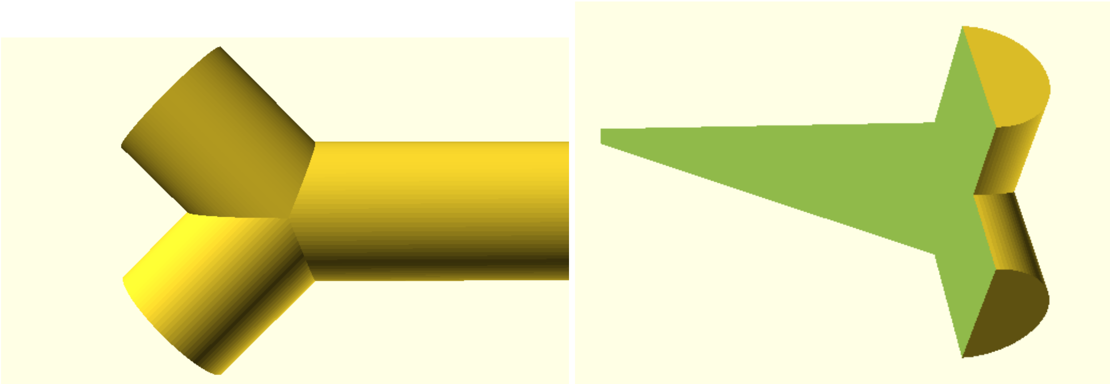

Figure 1. Geometry used longitudinally sectioned with two inlets.

stratified and the simulations of the flows patterns stratified with interface mixing.

The choice of mesh size to better represent two-phase flow is not a direct task, consideration should be given about the position of the interface zone between the phases, the regions close to the wall of the tube should be evaluated. The regions of refinement and the density of the mesh must be constructed knowing that the processing time which increases greatly with the number of cells in the mesh and many parameters are not correctly resolved with coarse meshes. During the work of Ávila (2016), the author performed mesh tests in order to identify the smallest number of nodes that presents a satisfactory result and lowest computational cost. For a pipe geometry of similar size to that used in this work, considering the pressure variation at two points, Ávila (2016) obtained an initial mesh with approximately 200000 nodes as more adequate, taking into account the precision and the computational cost. To evaluate the wave parameters, the author applied a much more refined mesh of 3200000 nodes, in order to reduce the computational cost caused by passing simulations from a coarse mesh to a refined mesh, the author used intermediary meshes. Based on the ideal number of nodes presented by Ávila (2016) to keep the computational cost low with good results and using intermediate meshes, the meshes used in this work were created.

To start mesh creation with the snappyHexMesh tool it needs to create a base mesh that encompasses pipe geometry. The base mesh created by the blockMesh tool has the maximum dimensions slightly larger than the pipe dimensions. The cells of the base mesh have the shape of hexahedrons (cubes) with all edges of the same value. The size of the edges of the cells of the base mesh used varies from 0.01 m for the initial coarse meshes and 0.005 m for intermediate mesh generation, thus starting with a more refined base mesh to be applied to snappyHexMesh tool processes.

With the exception of the initial coarse mesh, the meshes are generated with two refinement regions, one localized in the interface region between the phases and another region in the pipe wall, (Fig. 2). The refinement region at the phase interface is located throughout the pipeline and has easily adjustable settings so that it always encompasses the phase interface region. Due to its great length and importance for the simulation of the interface between the phases, it is expected that this region has the largest amount of volumes, especially in the simulation of SW cases with refined meshes, (Fig. 2 (d)).

The second refinement region located near the pipe wall follows the Layer settings in addition to the snappyHexMesh tool. Some basic conditions were established for the creation of this refinement region in the pipe wall, all meshes were generated with seven layers from the surface of the tube, the expansion ratio of these layers was defined as 1.2. The thickness of the seventh layer (the layer closest to the center of the tube), has its value fixed and varies from 0.002 m, for the coarse mesh used, and for the most refined mesh it was defined a value equal to 0,0002 m, both settings was used to the simulation of stratified smooth (SS) flow case.

### 3.2 Boundary conditions and pre-processing settings

For the simulation of the smooth stratified flow (SS) the multiphaseEulerFoam solver was used, this solver has the capacity to solve several phases being compressible or not. An Euler-Euler two-fluids approximation was used to identify the phases. The inner region of the pipeline at time 0 is initially configured to be filled only with the water phase, for the phase identification the alpha = 1 value was defined for the water phase and for the oil phase the alpha value was equal to 0. Each input region receives only a single phase, so no fractional alpha values are employed at the input, thus resulting in alpha.oil and alpha.water being obtained in the main tube.

The drag coefficient attributed to the simulations follows the model proposed by Schiller and Naumann (1935), in cases where the Reynolds number is greater than 1000, the value of the drag coefficient has a constant value of 0.44. Some studies use other values of drag coefficient, however, to date no studies have been published with significant new values for liquid-liquid flows in pipelines. Therefore, the choice of the drag coefficient was based on the work of Ávila (2016), on the recommendations in the ANSYS (2013) manual and on the frequent use of OpenFOAM tutorials, in order

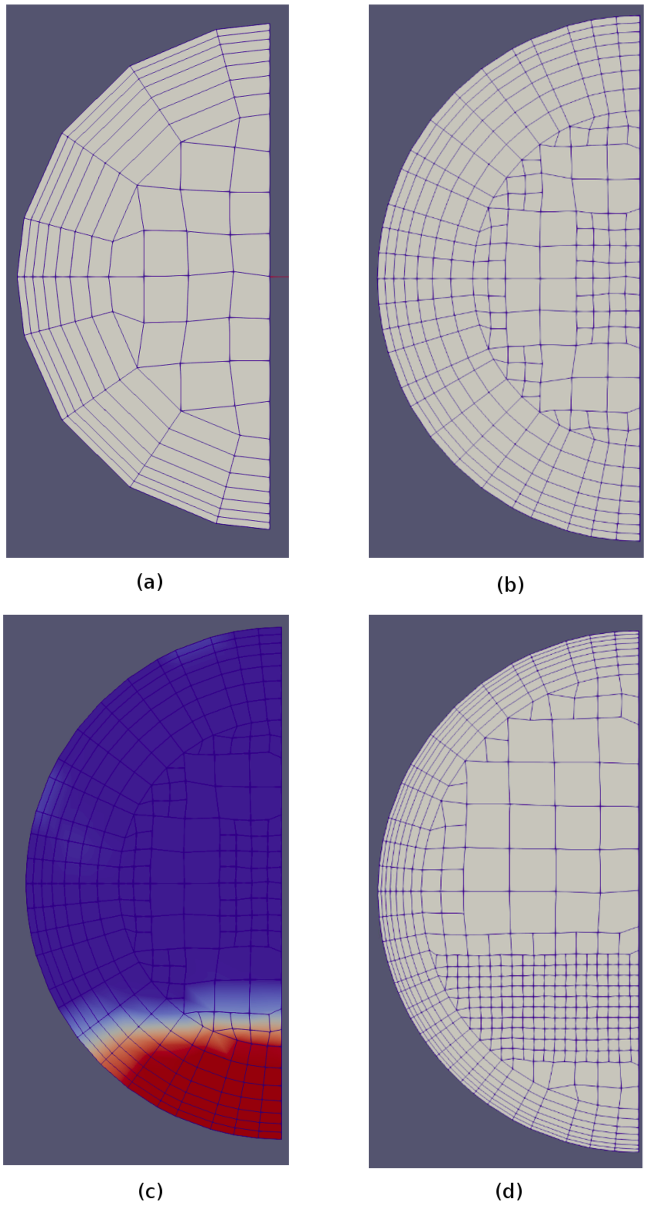


Figure 2. Variation of created meshes: (a) Coarse mesh for obtaining initial data; (b) intermediate mesh; (c) alpha water variation and interface between phases; (d) Refined mesh for the interface simulation of SW flow pattern

to use the model proposed by Schiller and Naumann present in the libraries of both softwares.

A no-slip boundary condition is imposed on the wall of the pipe, the symmetry condition is applied to the surfaces that represent the longitudinal cut in the pipe. Following data from Castro (2013), some sets of superficial velocities generate smooth stratified flow (SS) were identified, the combination of superficial velocities used in the simulation are Usw = 0.03 m/s and Uos = 0.02 m/s. In which, the surface velocity of the oil applied at the upper inlet of the pipe and the surface velocity of the water at the lower inlet, (Fig. 1). The contour conditions applied in the simulation of the smooth stratified flow pattern are reported in the Table 1.

Due to the dynamic behavior of two-phase flow, a transient simulation with a time step of 0.0001 s is performed, with the option of adjusting time step, in order to keep the Courant number below 0.9. Therefore, an implicit Euler scheme was selected for the time discretization and for the spatial discretization it was used the second order scheme of Van-Leer, which is a scheme that combines the system of limitations of Upwind and the precision of the differential-central scheme), thus being a stable and propitious method to produce correct physical results.

### 3.3 Initial results and discussion

Comparing visually the results obtained by the simulation of the smooth stratified flow pattern, a great similarity is observed in the images performed by Castro (2013), confirming the obtaining of the smooth stratified flow pattern,

Table 1. The boundary conditions used in the simulation of the smooth stratified flow (SS)

| Surfaces | alpha.oil | alpha.water | p_rgh | U.* |
|---|---|---|---|---|
| inlet1 | fixedValue | fixedValue | fixedFluxPressure | fixedValue |
| inlet2 | fixedValue | fixedValue | fixedFluxPressure | fixedValue |
| outlet | inletOutlet, inletValue=0 | inletOutlet, inletValue=1 | prghTotalPressure | pressureInletOutletVelocity |
| symmetryin | symmetry | symmetry | symmetry | symmetry |
| symmetrywall | symmetry | symmetry | symmetry | symmetry |
| tube | zeroGradient | zeroGradient | zeroGradient | noSlip |
| winlet1 | zeroGradient | zeroGradient | zeroGradient | noSlip |
| winlet2 | zeroGradient | zeroGradient | zeroGradient | noSlip |

(Fig. 3), (Fig. 4), where the black phase represents the oil phase and the white phase represents the water phase.

Figure 3. Experimental image of the smooth stratified flow pattern (SS) (Uws = 0.03 m/s, Uos=0.02 m/s).

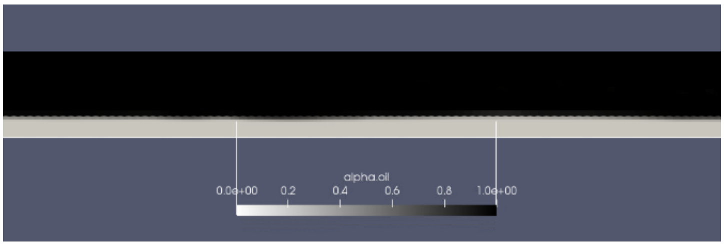


Figure 4. Numerical simulation image of the smooth stratified flow pattern (SS) (Uws = 0.03 m/s, Uos=0.02 m/s), using the intermediate mesh.

To perform the analysis of the parameters obtained by the simulation, a control volume was established, this volume is used to compare the values obtained by simulation with Castro's experimental data. The control volume is located between the cross sections of the 1.5 m and 1.75 m points measured from the start of the main pipe. These points can be used because in this section the phases are already developed. The values are taken from the time that the oil phase has traveled twice the length of the tube so that the simulation time varies according to the superficial velocity of the oil. Figure 5 demonstrates the variation of alpha.oil in the cross-section of the 1.75 m point and in the longitudinal section between the two sections.

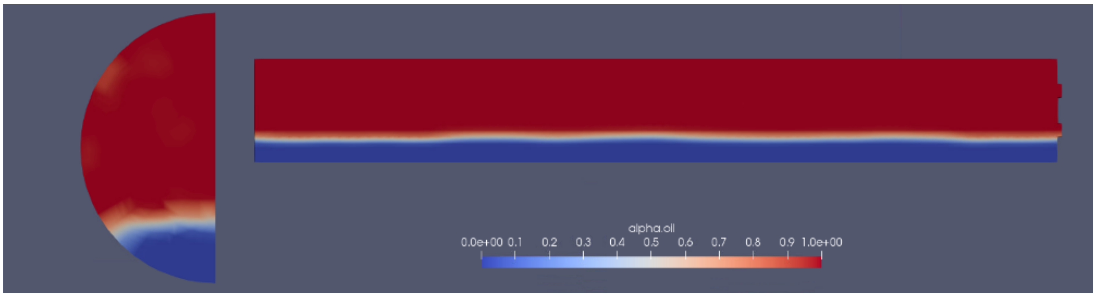


Figure 5. Variation of alpha.oil in the cross-section at 1.75 m and the longitudinal section between 1.5 m and 1.75 m.

The pressure gradient measurement is performed between the two cross-sectional points already identified. The Castro (2013) pressure drop data are presented as -dp/dz, in unit Pa/m. Thus, at a distance of 0.25 m between the two cross-sections of the control volume should be multiplied by four to have the pressure gradient per meter, this task can be performed since the pressure varies linearly with the length. The "Figure 6 (a)" shows the pressure variation by the distance of 0.25 m for the simulation of the case of smooth stratified flow pattern with Uos=0.02 m/s and Usw=0.03 m/s, obtained with the intermediate mesh, and the "Fig. 6 (b)" shows for the same case, the pressure variation at a distance of

0.25 m for the coarse mesh. Thus, the pressure gradient obtained by the simulation of the SS case with the intermediate mesh is 110.344 Pa/m and for the coarse mesh of 96.568 Pa/m, and the value of the pressure gradient obtained by Castro (2013) was 23.65 Pa/m. The difference between the values of the empirical data in relation to the simulated values is noticiable. The difference between the simulated pressure gradient and the empirical pressure gradient, although large, is expected, since similar differences in pressure gradient occurred during the study of Ávila (2016). Such simulations used concepts similar to those applied in this study, which shows a tendency towards the error in relation of the pressure gradient at two-phase liquid-liquid simulations. Finally, it is an encouragement to test new multiphase models, such as VOF, as well as different configurations of contour conditions and turbulence models.

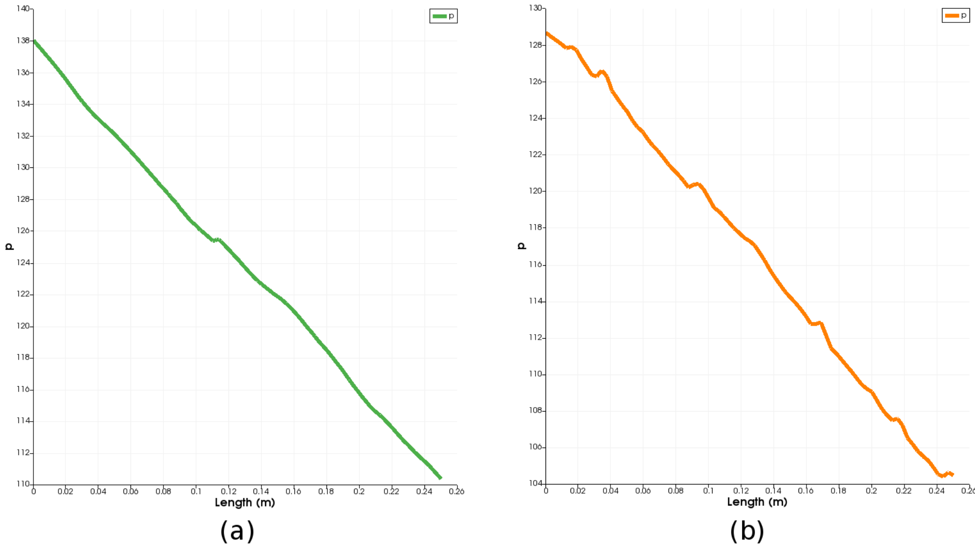


Figure 6. Geometry used longitudinally sectioned with two inlets.

In relation to the phase's fractions, the same procedure is performed, the mean values of phase's fraction are measured in the control volume located between the two cross-sections and then compared with the values obtained in the castro (2013) experiments. The value of the phase's fraction of water and oil is obtained with the intermediate mesh was alpha.water equal to 0.1907 and the alpha.oil is 0.8093, remaining stable and with values very close to them from 170 seconds of simulated flow. The phase's fraction obtained with the coarse mesh was 0.2460 for the alpha.water and 0.7539 for the alpha.oil, and different from the results obtained with the intermediate mesh, the results of the alpha are not stable between the seconds of simulated flow and have a considerable variation between the interpolations.

During the simulation of the smooth stratified flow pattern with Uso = 0.02 m/s and Usw = 0.03 m/s, different mesh configurations were tested, the coarse mesh being approximately 80000 elements and the intermediate mesh approximately 350000 elements. The pressure gradient results between the meshes were close. However, the visual result of the stratified flow pattern of the coarse mesh is much poorer than the result presented by the intermediary mesh. This trend was also represented by the variation of the phase's fractions between the two meshes. In order to identify interface parameters, a more refined mesh was elaborated with approximately 800000 elements, this mesh will also be applied to the simulation of the wavy stratified flow pattern. The simulation of this flow pattern (SW) would use the internalfield data obtained with the simulation of the (SS) pattern presented. As previously reported in SW simulation the aqueous phase will be considered as turbulent. Finally, in the simulation of this case it will be used the SST K-omega turbulence model, which is known to present the best of the k-epsilon and k-omega turbulence model.

## 4. CONCLUSIONS

A review was described covering studies of liquid-liquid two-phase flows using numerical methods, more specifically about CFD techniques. It was tried to describe common knowledge and currently techniques applied to solve such flows. With this, a brief description was realized about the turbulece and multiphase models commonly used by the authors. Although some authors have reported satisfactory results, some characteristics remain unresolved and doubts about the parameters and models used have not yet been fully answered. Therefore, it is still needed studies comparing empirical and simulated data, as well as more implementation of CFD techniques, in order to identify new and better parameters for horizontal and inclined liquid-liquid two-phase flows. Considering this work has been carried out to identify and propose simulation models and parameters used in CFD to simulate two-phase liquid-liquid flows in pipes. The initial simulations for the smooth stratified case were described briefly in this article, and preparations for simulation of cases of stratified wavy and stratified with interface mixing are being performed, and will be presented in future works.

## 5. ACKNOWLEDGEMENTS

The authors wish to thank the company PETROBRAS for their financial and technical support in this study. Also, we would like to thank ANP (National Agency of Petroleum, Natural Gas and Biofuels) for the support thourgh its "Compromisso de Investimentos com Pesquisa e Desenvolvimento". We would also like to thank the School of Mechanical Engineering (FEM) and the Center for Petroleum Studies (CEPETRO) both at the University of Campinas (UNICAMP). We would finally like to thank the ALFA research group for their support.

## 6. REFERENCES

Alagbe, S.O., 2012. *Experimental and numerical investigation of high viscosity oil-based multiphase flows*. phdthesis, Cranfield University Department of Offshore, Process and Energy Engineering, Cranfield.

Ali, Z., Dhanasekaran, P.C., Tucker, P.G., Watson, R. and Shahpar, S., 2017. "Optimal multi-block mesh generation for cfd". *International Journal of Computational Fluid Dynamics*, Vol. 31, pp. 195–213.

Andersson, B., Andersson, R., Häkansson, L., Mortensen, M., Sudiyo, R. and van Wachem, B., 2012. *Computational Fluid Dynamics for Engineers*. Cambridge University Press, Cambridge, 1st edition.

ANSYS, 2013. *ANSYS Fluent Theory Guide 15.0*. ANSYS.

Ávila, R.P., 2016. *Estudo fenomenológico e numérico do escoamento estratificado óleo-água ondulado e com mistura na interface*. phdthesis, Universidade de São Paulo, São Carlos.

Castro, M.S., 2013. *Fenômeno de transição espacial do escoamento óleo pesado-água no padrão estratificado*. Ph.D. thesis, Escola de Engenharia de São Carlos, Universidade de São Paulo, São Carlos. doi:10.11606/t.18.2013.tde-18092013-164235.

Desamala, A.B., Dasmahapatra, A.K. and Mandal, T.K., 2014. "Oil-water two-phase flow characteristics in horizontal pipeline - a comprehensive cfd study". *International Journal of Chemical, Molecular, Nuclear, Materials and Metallurgical Engineering*, Vol. 8, No. 4, pp. 360 – 364.

dos Santos, J.S.T., 2009. *Estudo Experimental e Numérico da Parafinação em Escoamento de Hidrocarbonetos em Dutos Produtores*. phdthesis, Universidade Federal do Rio de Janeiro, Rio de Janeiro.

Ghosh, S., Das, G. and Das, P.K., 2010. "Simulation of core annular downflow through cfd-a comprehensive study". *Chemical Engineering and Processing: Process Intensification*, Vol. 49, No. 11, pp. 1222 – 1228.

Gopala, V.R. and van Wachem, B.G., 2008. "Volume of fluid methods for immiscible-fluid and free-surface flows". *Chemical Engineering Journal*, Vol. 141, No. 1, pp. 204–221.

Greenshields, C.J., 2018. *OpenFOAM: User Guide version 6*. OpenFOAM Foundation Ltd., http://foam.sourceforge.net/docs/Guides-a4/OpenFOAMUserGuide-A4.pdf, 6th edition.

Maliska, C.R., 2004. *Transferência de Calor e Mecânica dos Fluidos Computacional*. LTC, Rio de Janeiro, 2nd edition.

Paladino, E.E., 2005. *Estudo do Escoamento Multifásico em Medidores de Vazão do tipo Pressão Diferencial*. phdthesis, Universidade Federal de Santa Catarina, Florianópolis.

Pereira, C.C., 2011. *Estudo experimental e modelagem do escoamento estratificado ondulado óleo-água*. mathesis, Escola de Engenharia de SÃčo Carlos, Universidade de São Paulo.

Ribeiro, D.C., 2012. *Modelagem e simulação do escoamento de sistemas multifásicos em reatores agitados mecanicamente*. phdthesis, Universidade Federal de Santa Catarina, Centro Tecnológico. Programa de Pós-Graduação em Engenharia Química, Florianópolis.

Rodriguez, O. and Baldani, L., 2012. "Prediction of pressure gradient and holdup in wavy stratified liquid-liquid inclined pipe flow". *Journal of Petroleum Science and Engineering*, Vol. 96-97, pp. 140–151.

Rzehak, R. and Kriebitzsch, S., 2015. "Multiphase cfd-simulation of bubbly pipe flow: A code comparison". *International Journal of Multiphase Flow*, Vol. 68, pp. 135 – 152. ISSN 0301-9322. doi:https://doi.org/10.1016/j.ijmultiphaseflow.2014.09.005. URL `http://www.sciencedirect.com/science/article/pii/S0301932214001670`.

Taitel, Y. and Dukler, A.E., 1976. "A model for predicting flow regime transitions in horizontal and near horizontal gas-liquid flow". *AIChE journal*, Vol. 22, No. 1, pp. 47–55.

Torres-Monzón, C.F., 2006. *Modeling of oil-water flow in horizontal and near horizontal pipes*. mathesis, University of Tulsa.

Wang, Z., Zhang, Q., Zeng, Q. and Wei, J., 2017. "A unified model of oil/water two-phase flow in the horizontal wellbore". *SPE Journal*, Vol. 22.

Zikanov, O., 2010. *Essential Computational Fluid Dynamics*. Wiley, Hoboken, 1st edition.

## 7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.