17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT-2018-0495
NUMERICAL ANALYSIS OF A LIQUID-GAS TWO-PHASE FLOW IN A DISTRIBUTION SYSTEM

Henrique K. Eidt
Carolina C. Rodrigues
Luiz H. M. Lino
Paulo H. D. dos Santos
Rigoberto E. M. Morales
Multiphase Flow Research Center (NUEM), Federal University of Technology - Paraná (UTFPR) – Rua Deputado Heitor Alencar Furtado 5000, Bloco N, CEP 81280-340, Curitiba, Brazil
hkeidt@gmail.com, carolcimarelli@gmail.com, luizhlino@hotmail.com, psantos@utfpr.edu.br, rmorales@utfpr.edu.br

Abstract *In the extraction and production of petroleum offshore some equipment are used to separate the liquid and gas phases of a two-phase flow in order to use electric submersible pumps. The existing centrifugal separators are generally oversized, which makes them harder to be installed and handled on the seabed. An alternative would be the use of smaller centrifugal separators. For that, it is necessary to develop a distribution system prior to smaller centrifugal separators. Thus, in this work, a numerical investigation of this distribution system is performed to analyze the division of the liquid and gas mass flow rates and the development of the liquid film flow. The two-phase flow gets into the distributor by the inlets, tangentially positioned, and a vertical ascendant flow liquid film is formed, influenced by the action of the centrifugal and gravitational fields. The performance of division is numerically analyzed regarding the mass flow rate in the outlets of the distribution system. For the numerical simulations, it was used the finite volume method based on finite element (ANSYS-CFX software 17.2), two fluids Eulerian-Eulerian inhomogeneous model and turbulence SST (Shear Stress Transport) model. For capturing the liquid-gas interface, the compressive advection discretization scheme is used. It was concluded that the distribution system provides high efficiency in divide the phases of the flow, and the length of the cyclonic chamber affects directly the equitable division of mass flow rate.*

Keywords: *two-phase flow, distribution system, numerical simulation, liquid film.*

1. INTRODUCTION

The extraction and production of petroleum offshore has increased over the years and because of that, improvements have been made to enhance the efficiency and optimized the equipment and processes involved in oil production. The path traveled by the oil, deep in the sea, between the wells to the platform, requires specific equipment to carry out the oil rig. Regularly, liquid-gas separation is done on the seabed to increase productivity and decrease production costs. The equipment used to separate these two-phases are called separators. Such devices are installed at the bottom of the sea, just after the wells exit, and then the phases are driven in different ways to the surface (Figure 1). For example, the liquid phase through an electric submersible pump and the gas rises by the density difference.

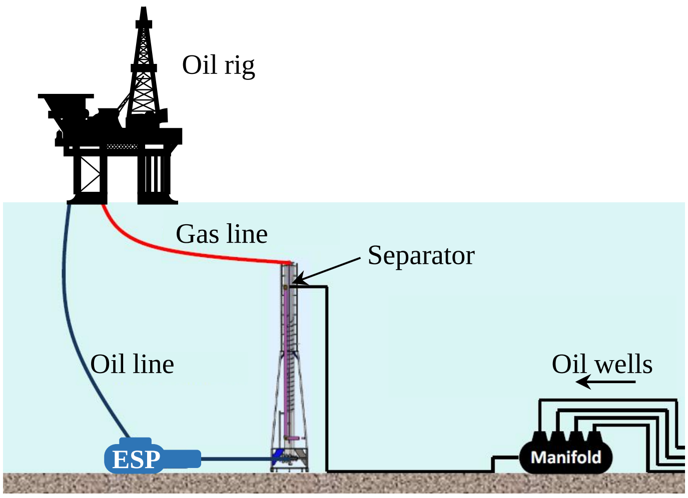


Figure 1. Schematic representation of the oil extraction process.

Separation on the seabed of the petroleum production usually occurs due to gravitational and centrifugal forces. Equipment such as Vertical Annular and Pumping System (VASPS) and Gas-Liquid Cylindrical Cyclone (GLCC) (Rosa *et al*., 2001) use this operating principle. These devices perform the separation process with high velocity and still have high separation efficiency when compared to traditional gravitational separators (Ninahuanca *et al.*, 2015). However, these separators have high dimensions, making it difficult to construct, install and maintain the equipment on the seabed. Thus, studies have been developed to devise a way to decrease the dimensions of the separators without affecting the separation efficiency and the oil productivity.

One way to solve this problem is to use a group of scaled-down separators. For this reason, an equipment capable of dividing liquid-gas flow approximately equitably into more separators is necessary and it is denominated distributor. The developed of this distributor also uses the centrifugal field concept, due to the tangential positioning of the inlets at the bottom as can be seen in Figure 2. After the flow go through the entrances, it develops in the cyclonic chamber under the action of the centrifugal field, forming an upward liquid film until reaching the exits. In this step, the combination of the tangential positioning of the outlets with the centrifugal flow, allows an equitable phase distribution.

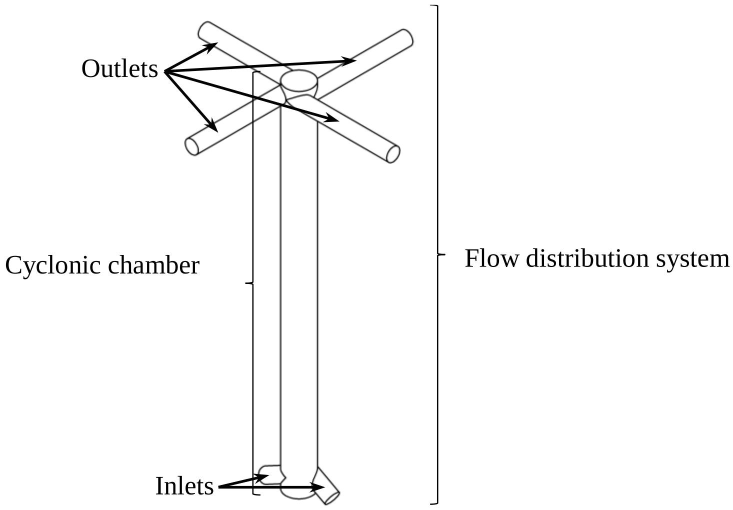


Figure 2. Schematic representation of the oil extraction process.

Recently some studies have been carried out on the distributor geometry and on the physical phenomena that are intrinsic in this type of flow. Eidt *et al*. (2016) initiated the study observing the liquid-gas behavior inside the distributor and the flow division in the four exits. This study was only numerical and they concluded that the distributor can perform an approximately equal flow division in the outlets. They also verified that as the flow rate develops in the cyclonic chamber, the liquid film becomes thicker and, consequently, the velocities decrease.

Ofuchi *et al*. (2017) developed an experimental bench to analyze the flow inside the cyclonic chamber. To analyze the gas fractions in the test sections, they used wire-mesh sensors. They positioned a 12x12 wire-mesh in the middle of the cyclone chamber and another one at the distributor entrance. Four tests were performed, varying the liquid and the gas velocities, all in slug flow. They observed that the elongated bubbles of the slug flow were broken and a liquid film was formed in the cyclonic chamber with high intensity because of the centrifugal field. In this way, more equitability can be obtained in the flow division without the presence of the intermittent flow.

Rodrigues *et al.* (2017) performed a numerical study with experimental validation of the single-phase flow inside the distributor cyclone chamber. They observed that single-phase flow requires high velocity so that the liquid film can be generated and maintained. However, they analyzed the transient effect of liquid film flow on the validation of the numerical model. Then, they analyzed the liquid division in the four exits of the distributor as a function of the cyclonic chamber length. They concluded that a cyclonic chamber with height between 6 and 12 diameters provides a more equitable flow division.

Eidt *et al*. (2017a) validated the numerical modeling with the results of Ofuchi *et al.* (2017) to consolidate the numerical model elaborated in software ANSYS 17.2. According to the comparison of the time average value of the liquid film thickness between the experimental, measured by the wire-mesh acquisition system, and the numerical one, the numerical model could be validated due to the convergence of the results.

Finally, Eidt *et al.* (2017b) performed, at the same liquid and gas flow rates of Eidt *et al.* (2017a), the numerical model validation with the aid of the ultrasound acquisition system. In this way, it was possible to compare the numerical results with the results of two experimental acquisition systems, consolidating the numerical model capable of representing the liquid-gas flow inside the distributor.

In this context, the present work has the purpose to investigate the flow behavior inside the distributor and the liquid and gas distribution in the four exits. The study focuses on the slug flow pattern, with dimensions enlarged twice (in scale), with similar entry conditions, and of 6 and 12 diameters length, 312 and 624 mm, respectively, for the cyclonic chamber.

2. NUMERICAL METHODOLOGY

Numerical simulations were performed for a water and air flow. The inlet condition was set within a range of superficial velocities of water (j_L) and air (j_G), which are defined, respectively, as:

$$j_L = \frac{Q_L}{A_P} \tag{1}$$

$$j_G = \frac{Q_G}{A_P} \tag{2}$$

where A_P is the cross section area of the pipe and Q is the volumetric flow rate of each phase. Values of both, j_L and j_G, were varied from 0.5 to 2.0 m/s.

The numerical model was developed and implemented using the ANSYS–CFX 17.2 software package. The flow inside of the cyclonic chamber has high turbulence, in consequence of this the shear stress transfer model (SST) was applied. It was also considered a transient, isothermal and incompressible flow. Due to the two-phase characteristic of the flow, an inhomogeneous Eulerian-Eulerian model was applied. This model is described by ANSYS (2015) and it is a mass weighted approach. Phases are considered as continua and interpenetrating. If non-drag forces and interfacial mass transfer between the phases are neglected, the governing equations for this case can be written as:

$$\frac{\partial}{\partial t}\left(r_i\rho_i\right) + \nabla\bullet\left(r_i\rho_i\vec{V}_i\right) = 0 \tag{3}$$

$$\frac{\partial}{\partial t}\left(r_i\rho_i\vec{V}_i\right) + \nabla\bullet\left(r_i\rho_i\vec{V}_i\times\vec{V}_i\right) = \nabla\left(r_i\mu_i\left(\nabla\vec{V}_i\times\left(\nabla\vec{V}_i\right)_T\right)\right) - r_i\nabla p + r_i\rho_i\vec{g} + \vec{F}_D \tag{4}$$

where V_i is the velocity of the phase i, r_i is the volume fraction of the phase, ρ_i is the density, μ_i the viscosity, ∇p the pressure gradient, g the gravity acceleration and F_D the drag force between the phases.

For the numeric model some simplifying assumptions regarding on the flow were required, such as:

i. Water was used for liquid phase and air for gas phase for numerical simulation;
ii. The initial condition considered was the geometry with only liquid phase and velocity equal zero;
iii. Isothermal (300 K) and no-slip wall boundary conditions;
iv. Turbulent intensity equals to 5%, which is equivalent to a viscosity ratio equals to $\nu_T/\nu = 10$;
v. Opening condition at outlets, pressure difference between inside the cyclonic chamber and outside is equal to zero, $\Delta P = 0$;
vi. To determine the location of the liquid-gas interface is adopting a compressive discretization scheme;
vii. Slug flow were set at the inlets, and one representative unit cell for each case was estimated.
viii. Hybrid mesh were used for the model with tetrahedral volume mesh at the inlets region and hexahedral volume mesh at the cyclonic chamber and it was made a refinement in the mesh near the walls as required by the turbulence model SST, as shown at the Figure 3.

In the numerical mesh test was analyzed the variation of the liquid film thickness along the time, and comparing the results between four different types of mesh. After analysis, it was define that the second most refined mesh was capable to perform the liquid-gas flow without any interference from the mesh. Along these lines, the total simulation time necessary to reproduce the flow is 1 s, with a time step of 0.002 s.

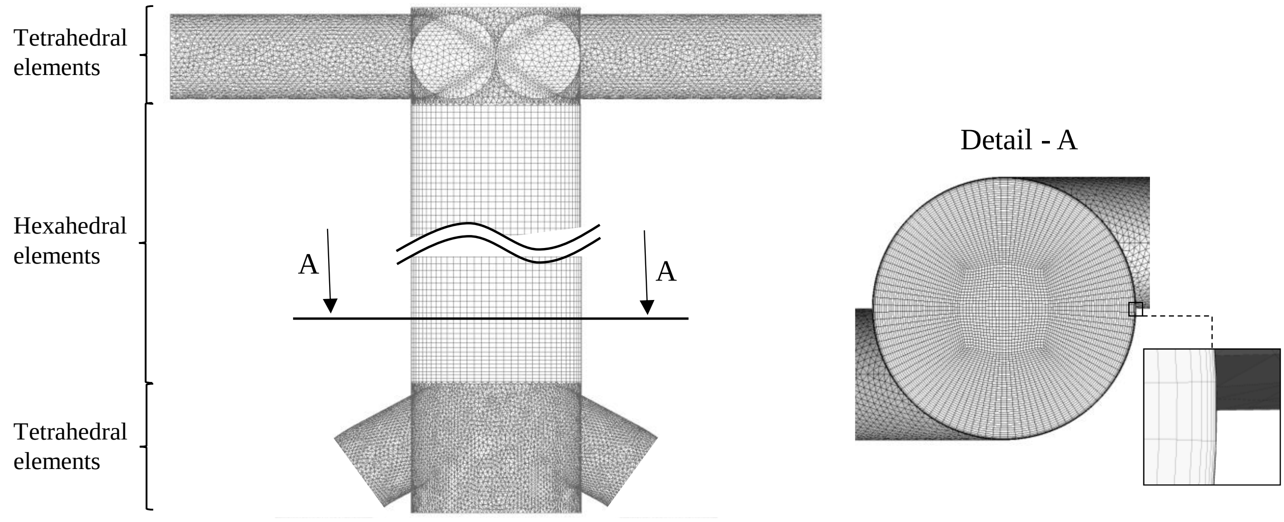


Figure 3. Numerical mesh used with tetrahedral and hexahedral elements.

3. RESULTS AND DISCUSSION

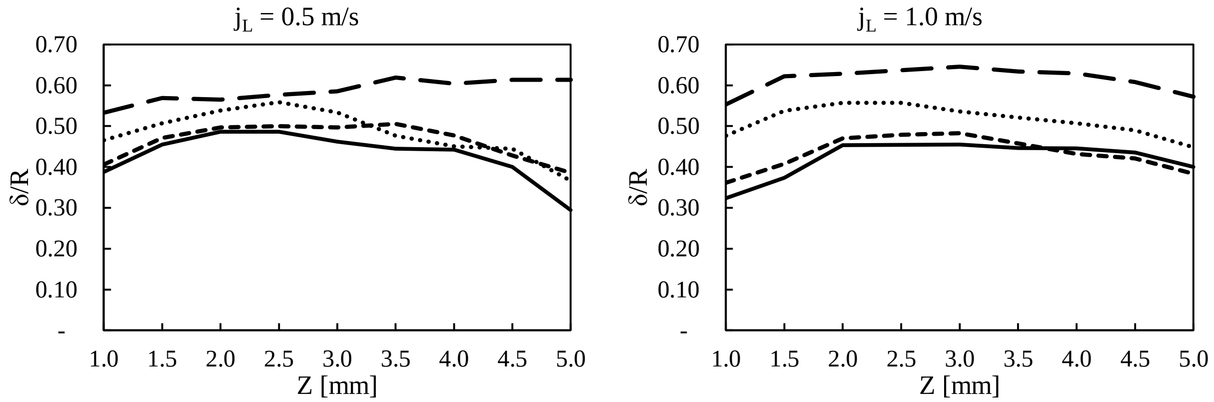
In order to analyze the liquid film flow inside the cyclonic chamber of the distributor system, it was made 16 simulations, varying the liquid and gas superficial velocities, as can be seen in the Table 1.

Table 1. Simulation cases.

Cases	***1***	***2***	***3***	***4***	***5***	***6***	***7***	***8***	***9***	***10***	***11***	***12***	***13***	***14***	***15***	***16***
j_L ***[m/s]***	0.5	0.5	0.5	0.5	1.0	1.0	1.0	1.0	1.5	1.5	1.5	1.5	2.0	2.0	2.0	2.0
j_G ***[m/s]***	0.5	1.0	1.5	2.0	0.5	1.0	1.5	2.0	0.5	1.0	1.5	2.0	0.5	1.0	1.5	2.0

Thereby, the analysis of the liquid-gas flow behavior begin with the variation of the thickness liquid film flow along of the length of the cyclonic chamber, showed in Figure 4, for the geometry with six diameter of length. As can be seen, it was made a graph for each group of superficial liquid velocity to observe the influence that the gas can cause in the flow. It is remarkable that with the increasing in the mass flow of the gas, the thickness of the liquid film flow decrease.

It is also noticeable that for high liquid velocities (j_L=1.5 m/s and j_L=2.0 m/s) the liquid film thickness decreases to a minimum value at around Z=1.5D and starts to increase again. In some cases (j_L=1.5 m/s, j_G=1.5 m/s; j_L=1.5 m/s, j_G=2.0 m/s; j_L=2.0 m/s, j_G=0.5 m/s and j_L=2.0 m/s, j_G=1.0 m/s;) the film thickness, after reaching the minimum value, increases to a maximum value and slowly decreases again, describing, roughly, a wave shaped graph. It is believed that this wave pattern occurs due to centrifugal forces and a phenomenon known as overlap. This event is when streamlines with different inclinations happens to cross each other, increasing the liquid film thickness at the crossing point. As the liquid flow increases, the flow becomes more intense and, consequently, the centrifugal field increases, reducing the liquid film thickness, by generating a pressure gradient in radial direction, and generates secondary flows, with different inclinations, causing the streamlines to overlap further in the cyclonic chamber (Morandin, 1999).



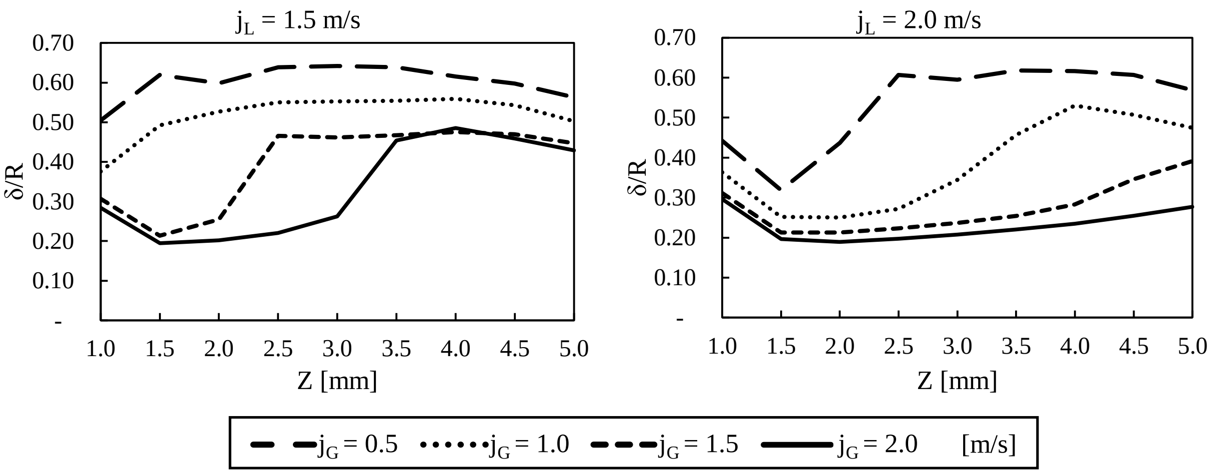


Figure 4. Variation of the thickness liquid film flow over the length of the cyclonic chamber with six diameters of height.

In the same way, Figure 5 shows the variation of the thickness liquid film flow over the length of the cyclonic chamber, for the geometry with 12 diameter of length. It is observed that, equally, to the lower geometry, lower superficial velocities of liquid presents a decrease in the thickness of the liquid film with the increase of the length of the cyclonic chamber. Moreover, differently from the geometry with six-diameter length, the wave pattern is intensified for the liquid superficial velocity of 2.0 m/s. This happens because 312 mm is not enough for the wave pattern develop with a j_L=2.0 m/s. This suggests that the higher the flow rate, the higher is the length required to develop the wave pattern.

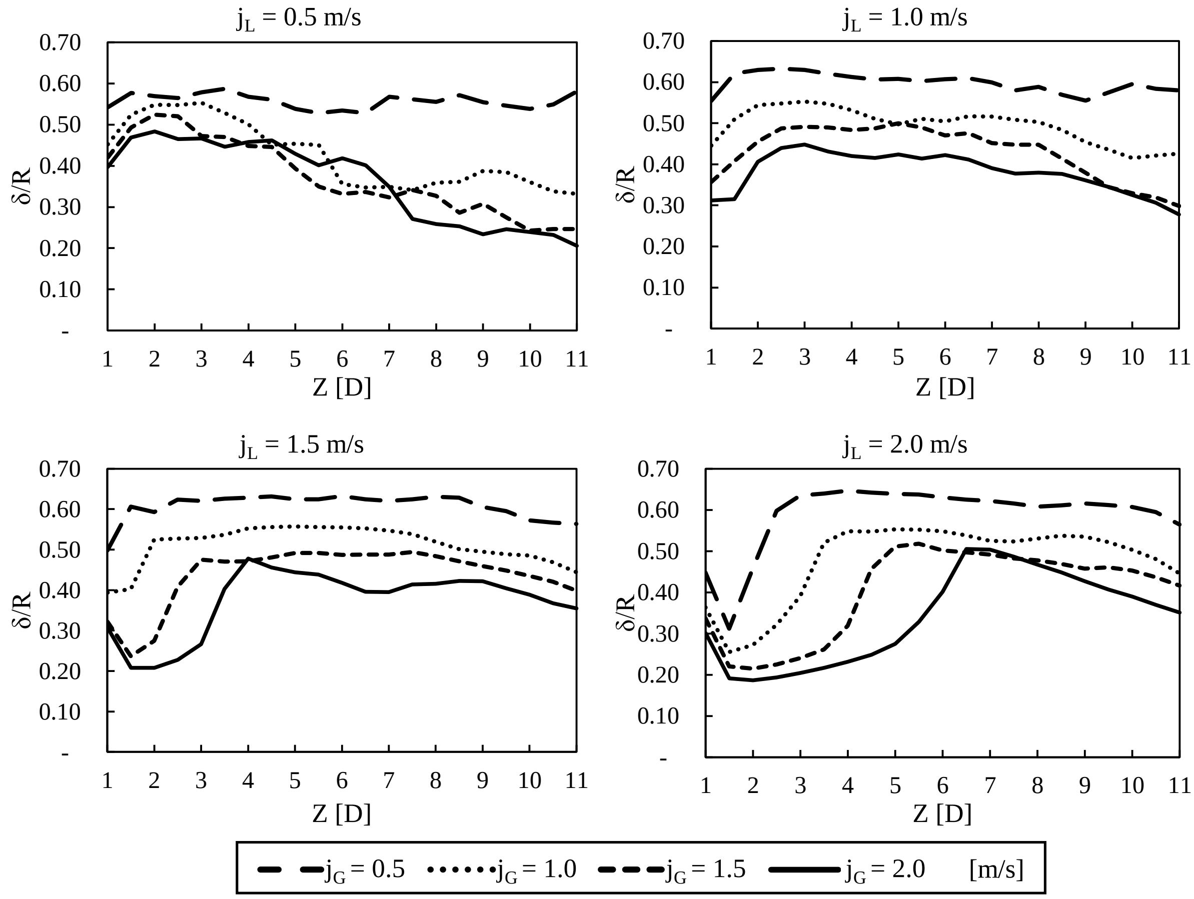
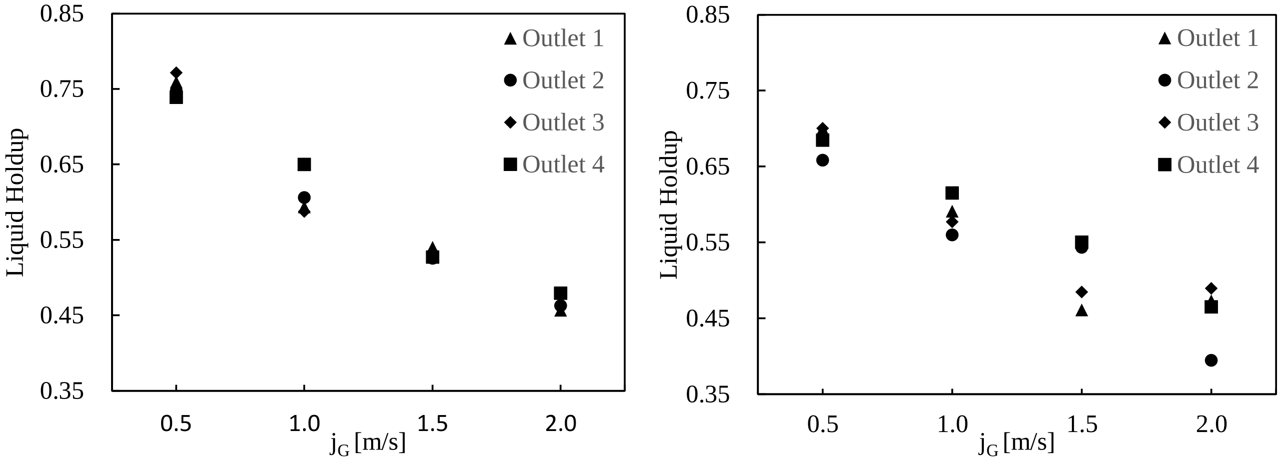


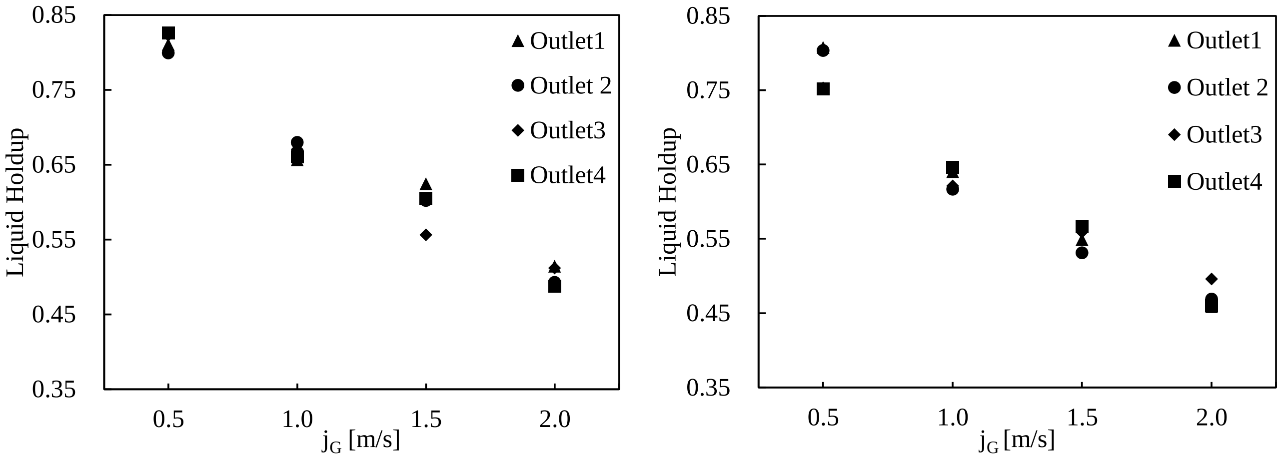
Figure 5. Variation of the thickness liquid film flow over the length of the cyclonic chamber with 12 diameters of height.

Figure 6 and Figure 7 shows the liquid holdup for each one of the four outlets for the six-diameter geometry and the twelve-diameter geometry, respectively. It is noticeable, by comparing both figures with Figure 4 and Figure 5, that the so-called wave pattern has a positive influence at the equitability of distribution. For example, at Figure 6 (a), where the wave pattern is developed, the liquid holdup at each outlet is satisfactory equitable. At Figure 6 (b), for the same gas superficial velocities, the wave pattern is not developed and it is possible to note a less equitable distribution. However, increasing the length of cyclonic chamber to 624 mm, extension enough for the wave pattern develops, in the same cases, the equitability of the distribution is increased. Even though the equitability is satisfactory for all cases, this suggests that a developed wave pattern is desirable to achieve an equitable distribution at the outlets.



(a) (b)

Figure 6. Average liquid holdup at each one of the four outlets for the geometry with six-diameter height. (a) For j_L=1.5 m/s and; (b) for j_L=2.0 m/s.



(a) (b)

Figure 7. Average liquid holdup at each one of the four outlets for the geometry with 12-diameter height. (a) For j_L=1.5 m/s and; (b) for j_L=2.0 m/s.

Figure 8 reveal the mean behavior of the liquid film flow inside the cyclonic chamber for the two geometries (6 and 12 diameter length). As it can be noticed, the average flow performance is similar in the two geometries in every case showed. This imply that the increase in cyclonic chamber length does not change the behavior of the flow in the beginning of the cyclonic chamber.

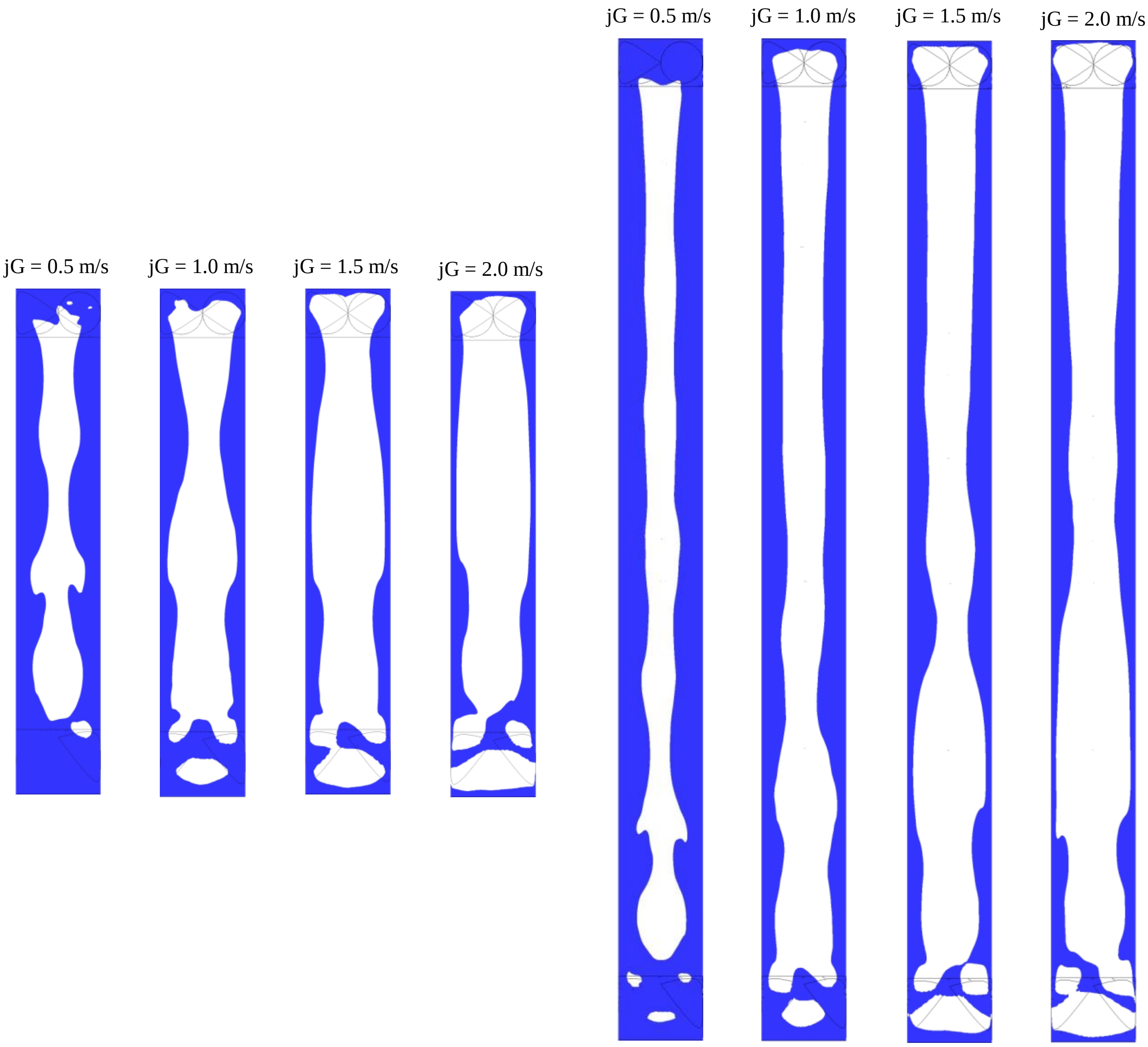


Figure 8. Average in time of the thickness liquid film flow for the cyclonic chamber with 6 and 12 diameter, for the superficial velocity of the liquid of 2.0 m/s.

Finally, it is possible to see the variation of the mass flow rate in the four outlets of the distribution system along the time, for the case of j_L=1.5 m/s and j_G=1.5 m/s, in the Figure 9. The graphic shows the relative mass flow rate for the time plus 0.4 s of the flow. It is noticed, that with the increase of the cyclonic chamber length decreases the division equitable of the phase. This is due to the flow need to go through a larger path until reach the four outlets. In this way, the liquid is for more time under the action of the gravitational field and the frictional force that the wall exerts in the phase, decreasing the liquid velocity and, consequently, increasing the flow instability. Even with these differences in the mass flow rate along the time, the maximum difference between the four outlets of the average liquid flow rate is lower than 15%. Representing, in the worst case scenario, relatively equivalent values of liquid and gas flows in the four outlets

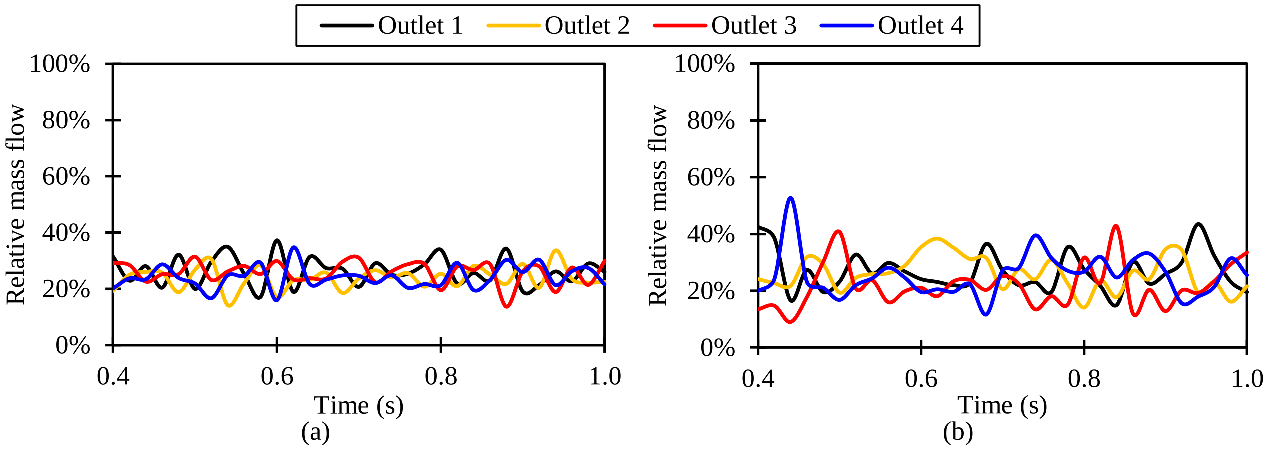


Figure 9. Variation of the liquid mass flow rate along the time with velocities j_L=1.5 m/s and j_G=1.5 m/s to: (a) 6 diameters and (b) 12 diameters length.

4. CONCLUSIONS

This paper presents numerical study of the behavior of an ascendant liquid film flow under the effect of centrifugal and gravitational fields inside the cyclonic chamber of a distribution system. It was also analyzed the division phase at the distributor outlets, the behavior and average thickness liquid film flow inside the cyclone chamber are the purpose of the present study.

From the study concerning the liquid film behavior in the cyclone chamber, it may be affirmed that the average thickness increases with the decreasing of the mass flow rate of the gas. In other words, the increase of the liquid mass flow rate implies in an intense centrifugal field and the occurrence of a wave pattern that helps the equitable division of the phases. In addition, it is possible to conclude that the cyclonic chamber length affects directly in the distribution efficiency of the phases in the equipment, in which the distributor having six length diameters is sufficient to evenly distribute the liquid mass flow rate for that range of superficial velocities. However, more study is required to define the exactly cyclonic chamber dimension that provides the better distribution.

5. ACKNOWLEDGEMENTS

The authors acknowledge the carrying out of this work to the Multiphase Flow Research Center (NUEM), Federal University of Technology - Paraná (UTFPR) and PETROBRAS for the infrastructure, support and assistance for the development of numerical study of the upward liquid film flow under the effect of centrifugal and gravitational fields.

6. REFERENCES

César Y. O., Eidt, H. K., Santos, P. H. D. dos, Silva, M. J. da, Neves Jr., F., Morales, R. E. M., 2017, "Void Fraction Characterization of a Gas-Liquid Intermittent Flow Inside a Cyclonic Chamber", 9th World Conference on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics, June, Foz do Iguaçu, Paraná, Brazil.

Eidt, H. K., Rodrigues, C. C., Dunaiski, R., Ofuchi, C. Y., Santos, P. H. D. dos, Neves Jr., F., Morales, R. E. M., 2017a, "Numerical and Experimental Analysis of Vertical Ascendant Liquidgas Flow under Action of Centrifugal and Gravitational Fields", ASME 2017 - Fluids Engineering Division Summer Meeting, Waikoloa, Hawaii, USA.

Eidt, H. K., Rodrigues, C. C., Dunaiski, R., Ofuchi, C. Y., Santos, P. H. D. dos, Neves Jr., F., Morales, R. E. M., 2017b, "Experimental Validation by Ultrasonic Technique of the Numerical Analysis of an Ascendant Flow Under the Influence of the Centrifugal and Gravitational Fields", 24th ABCM International Congress of Mechanical Engineering, December, Curitiba, Paraná, Brazil.

Eidt, H. K., Rodrigues, C. C., Santos, P. H. D. dos, Morales, R. E. M., 2016, "Numerical Analysis of a Vertical Ascendant Two-Phase Flow in a Cyclonic Chamber Under the Action of Centrifugal and Gravitational Fields", 16th Brazilian Congress of Thermal Sciences and Engineering, November, Vitória, Espírito Santo, Brazil.

Morandin, M. L. et al., 1999. Modelagem de um filme liquido sob a ação combinada dos campos centrifugo e gravitacional de forças: hidrociclones. Campinas, São Paulo, Brazil.

Ninahuanca, H. M.; Azevedo, H. S., 2015. "Characterization of the Liquid Film Flow in a Centrifugal Separator", IV Journeys in Multiphase Flows, Campinas, São Paulo, Brasil.

Rodrigues, C. C., Eidt, H. K., Dunaiski, R., Ofuchi, C. Y., Santos, P. H. D. dos, Neves Jr., F., Morales, R. E. M., 2017, "Numerical Study of Transient Flow and The Influence of Height and Viscosity in a Cyclonic Chamber in a Distribution System", ASME 2017 - Fluids Engineering Division Summer Meeting, Waikoloa, Hawaii, USA.

Rosa, E., França, F. and Ribeiro, G., 2001. "The Cyclone Gas–Liquid Separator: Operation and Mechanistic Modeling", Journal of Petroleum Science and Engineering, Brasil.

7. RESPONSIBILITY NOTICE

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