



# Comparative Analysis of Multiphase Flow in a T Type Micro Junction

Gabriel M. Guimarães<sup>(✉)</sup> , Kevin L. Pinto , and Raquel J. Lobosco 

Federal University of Rio de Janeiro, Rio de Janeiro, Brazil

rlobosco@macae.ufrj.br

**Abstract.** This scientific research investigates the behavior of two incompressible fluids in a micro-T-junction channel by means of a computational simulation through the physical properties of water and air and the discretization of conservation equations for the dispersed phases. It was possible to represent different regimes of droplets formation in the computational domain. OpenFOAM version 4.0.1 software is an open platform used in the academic environment for fluid dynamic simulations and was chosen to perform the flow's numerical simulations due to its great acceptance in the scientific environment. The InterFOAM solver using the VOF method (Volume Of Fluid) developed for computational simulations of multi-phase flows was adopted in the simulations studied here. This study develops a comparative analysis of the droplet's formation behavior at the injection time and evaluates the flow aspect of the dispersed phase through the continuous phase in the main channel. From the variations in the flow rates, it was possible to verify the flow phases' dependence in determining the flow regime. This research was developed based on a literature review and evaluates the bubble injection shape and droplet coalescence in a T-junction microchannel. The results were compared with the literature data.

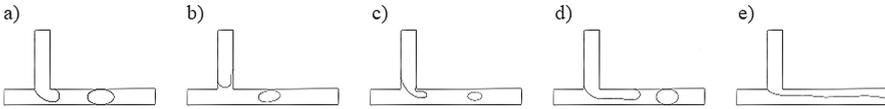
**Keywords:** Multiphase · Micro-fluidic · OpenFoam

## 1 Introduction

The multi-phase flow behavior has several characteristics depending on the conditions in which they are applied. In observing the flow of two incompressible fluids, with a constant density, in a microchannel, different flow regimes can be predicted depending on the speed rate, as showed in Fig. 1.

T-shaped joints are widely used for industrial cases of injection of a fluid and are frequently used to study the formation of drops in microchannels [1]. In this configuration, the continuous phase flows through the main channel, and the dispersed phase is injected, causing the formation of the droplets.

In a multi-phase microchannel flow, the fluid transport regime is characterized by the number of capillarities, shear stress, flow rate, and viscosity rate [2]. Among the various profiles already observed for multi-phase flows in microchannels in type T joints, five stand out, of which they are considered main as described below.



**Fig. 1.** Different types of flow observed. They are classified into five main types (a) Squeezing, (b) Transition, (c) Dripping, (d) Jetting and (e) Parallel Flow. Illustration of the numerical simulation at the OpenFoam software.

1. In the Squeezing regime, the drop fills the entire section of the main channel. The dispersed phase blocks the continuous phase in the channel for a brief moment, causing the drop before it detaches to gain a length, giving it a flow format.
2. In the Transition regime, the dispersed phase partially blocks the channel, forming drops with a flow rate, but with a shorter length than the Squeezing drops.
3. In the Dripping regime, the drops are spherical in shape and do not fill the entire channel. The point of separation of the drops is closer to the point of junction of the channels.
4. In the Jetting regime, part of the dispersed phase takes over the main channel with an elongated shape, during which the two immiscible fluids flow in parallel. The droplets detach themselves away from the junction point.
5. The Parallel flow profile has a particularity because it does not generate drops. The two fluids start to flow simultaneously in parallel through the main channel without any mixing of the fluids [3].

## 2 Modeling

The parameters used to perform the numerical analysis in the OpenFOAM open software version 4.0.1 is described below. The numerical results attend to be mass and momentum conservation for two incompressible flows.

### 2.1 Geometry of the Flow Channel

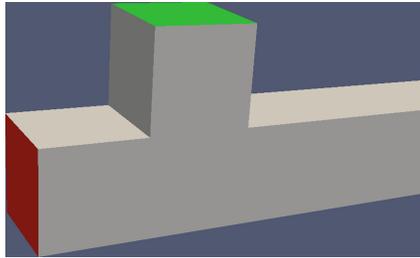
To validate the numerical results of the computational model, a geometry similar to the study carried out by S. Arias [4], as shown in Fig. 2. The geometry differs only in the length of the main channel, which in the study carried out here, is twice as long as the one studied by S. Arias, totaling 20 mm, in order to make it possible to analyze the development of drops in a computational domain.



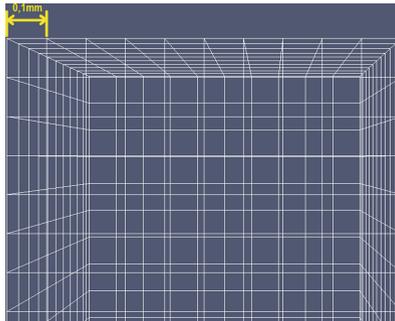
**Fig. 2.** T-junction dimensions of the geometry.

The T-type joint has two inlet and one outlet faces. The phases have identical input characteristics differentiated only by the fluid injection direction. The liquid phase has a horizontal entrance while the gas enters in the vertical direction. As in microchannel studies, the gravitational forces are considered null and the phase input direction doesn't affect the final results [5].

The mesh was defined to distribute the entire geometry in cells of the same size and shape, aiming to separate all geometry in symmetrical cubic unit cells with a 0.1 mm edge in a hexahedral mesh. The mesh characteristics are very important for capturing the interface in a multi-phase flow because of the interface capturing numerical challenges. Figure 3 illustrates the inlet boundary conditions. The gas inlet is at the green surface, and the liquid inlet is at the red face. The cell mesh is illustrated in Fig. 4.



**Fig. 3.** Entry Faces - red for water and green for air.



**Fig. 4.** Unit cell detail.

## 2.2 Boundary Conditions

Some boundary conditions are defined in order to simplify and make flow analysis possible.

To perform the study through a computational analysis, the OpenFOAM solver called *interFoam* was used. That has the multi-phase simulation capabilities for two incompressible fluids. This solver performs the studies following the methodology of interfacial treatment between volumetric fractions (VOF; Volume of fluids). Regardless of

the application in OpenFOAM, the fluid volume method has been shown to be quite effective and robust in the treatment of multi-phase flows in several platforms [6].

All the possible parameters to be set as constants were thus made, with the water and air properties, so that the analyzes depended only on the variation of the fluid inlet speed to change the flow profiles.

The geometry faces are defined in four different groups where the boundary conditions are set. Each of those groups has a different characteristic, and they are defined as Wall, Gas inlet, Liquid inlet, and Outlet, as shown in Figs. 2 and 3.

The group of wall surfaces makes the flow an internal flow. They were defined with the adhesion principle and without slip.

Input1 face is defined as a continuous velocity flow, and the called face input2 has characteristics similar to input1. The only difference is the normal direction and is defined as a fixed velocity for the gas.

Finally, the outlet face, towards which the flow of the phases is directed. This face has interaction with the external environment, making it necessary to consider external atmospheric pressure.

### 2.3 Flow Speed Variation

Once the parameters mentioned above are fixed, the flow properties will depend only on the variations of the phases' input velocities. Based on studies of the same case, a range of speeds was adopted in which it is expected to obtain comparative results [4].

At first, we seek to analyze the formation of two distinct flow profiles, the dripped and the slug. For this, speeds between 0.106 m/s and 0.531 m/s were adopted for the continuous phase, and, for the dispersed phase, a variation between 0.08 m/s and 0.34 m/s.

For the second part of the analysis, three speeds were tested for each phase. Continuous phase with 0.531 m/s, 0.318 m/s and 0.106 m/s and dispersed phase of 0.08 m/s, 0.24 m/s and 0.34 m/s in order to treat with these data a drop formation map and analysis of the frequency of drop formation.

## 3 Results and Discussions

The results were carried out to determine the development between the main flow profiles analyzed in experimental studies of a similar analysis [4]. Three speeds were defined for the continuous phase, and, for each of them, a combination with three different speeds for the dispersed phase was conducted. These analyzes generated a total of nine analyzed cases. In addition, two other cases for specific velocities were conducted in order to compare the flow profiles found with the one carried out by [4].

### 3.1 Droplet Formation

The characteristics of the dispersed phase injection process happen following the sequence, as shown in Fig. 5.

The images show the progress of the droplet formation in the Slug flow regime. The characteristics of the bubble formation was recently defined by [2].

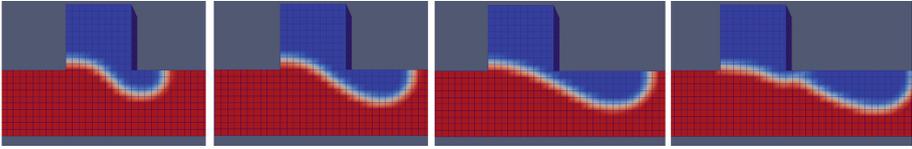


Fig. 5. Droplet formation

### 3.2 Flow Profiles

In order to evaluate the flow regimes, numerous simulations were carried out to characterize the drip and slug flow profiles. The inlet boundary conditions were varied out in the intervals between the maximum and minimum gas and Liquid velocity ( $V_G$  and  $V_L$ , respectively), as shown in Table 1.

**Dripping Flow.** The dripping flow occurred with speeds  $V_L = 0.531$  m/s and  $V_G = 0.08$  m/s generating the following flow profile, as shown in Fig. 6. This profile is defined by the semi-spherical shape that the drop presents, the high frequency of drop formation and the partial filling of the main channel. The dispersed phase takes the upper part of the channel and, as noted, the release time of the drops is 5 ms.

Comparing the Arias [4] experiment, the Dripping flow happens with the same speed values of phases.

**Slug Flow. Slug Flow Occurs with Velocities  $V_L = 0.350$  m/s and  $V_G = 0.242$  m/s.** This flow profile is characterized by the elongated shape that the drop takes, less spacing between drops, and partial obstruction of the main channel, as shown in Fig. 7. Comparing with the study by Arias and D. Legendre [4] to obtain the same drop profile, it was necessary to vary the velocity of the dispersed phase.

### 3.3 Droplets Formation Frequency

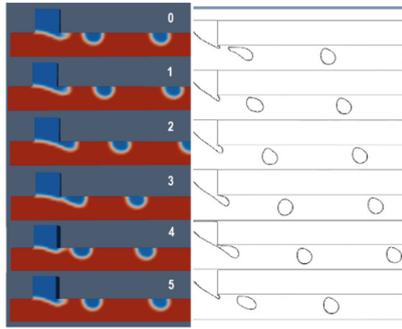
As a means of standardizing the data reading, flows were analyzed from the moment the first drop reaches the end of the main channel. Nine simulations were carried out combining three speeds for the continuous phase (0.106 m/s, 0.318 m/s and 0.531 m/s) and another three for the dispersed phase (0.08 m/s, 0.24 m/s and 0.34 m/s). The period  $T$  in seconds read for each case is described in Table 2.

The data from Table 2 of frequencies for the simulations performed in OpenFOAM were plotted in the graph shown in Fig. 8 along with the lines taken from the study by Arias and D. Legendre [4] that refer to the frequencies of drop formation in controlled experiments in the laboratory. Most of the data obtained in the simulations were satisfactory in relation to the laboratory study. Except when the velocity of the dispersed phase of 0.24 m/s is simulated combined with the velocities of the dispersed phase at 0.318 and 0.106, the frequencies found were slightly above the expected.

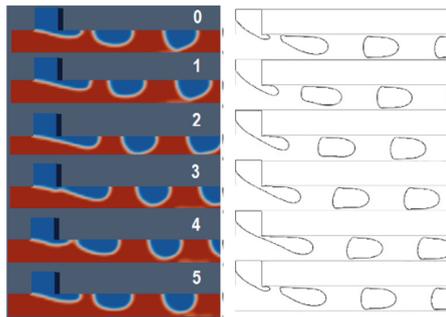
Despite the divergence in the frequencies of two of the speed combinations, the droplet formation pattern in the simulations responded well to the result obtained in the laboratory by [4].

**Table 1.** Interval between maximum and minimum phase speeds.

	Máx.	Min.
-		
$V_L$ (m/s)	0.531	0.318
$V_G$ (m/s)	0.242	0.068



**Fig. 6.** Comparison between simulation performed in OpenFOAM 4.0.1 and bubbles shape experiments performed by Arias and D. Legendre [4].

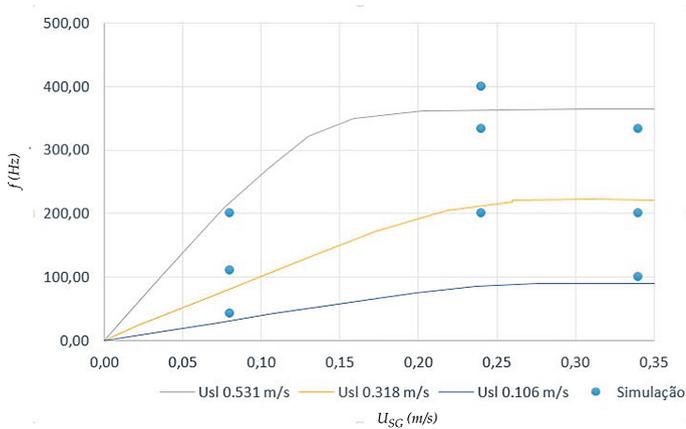


**Fig. 7.** Comparison between simulation performed in OpenFOAM 4.0.1 and the bubbles profile performed by Arias and D. Legendre [4].

**Table 2.** Droplet generation frequency.

$f(\text{Hz})$	$V_G = 0.08 \text{ m/s}$	$V_G = 0.08 \text{ m/s}$	$V_G = 0.08 \text{ m/s}$
$V_L = 0.106 \text{ m/s}$	43.48	333.33	100
$V_L = 0.106 \text{ m/s}$	111.11	200	200
$V_L = 0.106 \text{ m/s}$	0.003	0.0025	0.003

The frequencies obtained in the flow profiles analyzed show that there is a well-defined pattern of droplet formation for each flow rate of the continuous phase, that the increased injection speed of the dispersed phase makes the pattern of droplet formation stable, and the frequency of formation will be the same for higher injection speeds of the dispersed phase.



**Fig. 8.** Comparison between frequencies simulated in OpenFOAM and graph generated in laboratory analysis. Source: Arias and D. Legendre [4].

## 4 Conclusions

The purpose of this scientific research was to perform in OpenFOAM the multi-phase flow in a microchannel at T junction and to evaluate the aspects of the drip and Slug flow profiles based on the laboratory analysis performed by [4].

Through the computer simulations, it was possible to obtain a satisfactory result for the bubble formation profiles.

As it is a flow of micro fluids, the parameters of initial conditions, boundary conditions, geometric properties, fluid, and flow properties must be well defined in order to minimize numerical errors and simulation failure.

The InterFOAM solver used the Volume of Fluid method (VOF) and showed capabilities to reproduce the interfacial effects in the tested regimes in a microscale.

New analyses still need to be carried out to better understand the bubble coalescence phenomena.

## References

1. Leman M (2015) Microfluidique en gouttes à l'échelle femtolitrique. L'Université Pierre et Marie Curie, Paris
2. Malekzadeh S, Roohi E (2015) Investigation of different droplet formation regimes in a t-junction microchannel using the vof technique in openfoam. Springer, Media Dordrecht, Mashhad, Iran, p 13
3. Guillot P, Colin A (2005) Stability of parallel flows in a microchannel after a t junction. The American Physical Society, Pessac, p 4
4. Arias S, Legendre D, González-Cinca R (2012) Numerical simulation of bubble generation in a t-junction. Institut de Mécanique des Fluides de Toulouse (IMFT), Toulouse, França, p 60

5. Qian D, Lawal A (2010) Numerical study on gas and liquid slugs for Taylor flow in a T-junction microchannel. *Chemical Engineering Science*, Hoboken, USA, p. 17, 2006. Author, F.: Contribution title. In: 9th international proceedings on proceedings, Publisher, Location, pp 1–2
6. Deshpande SS, Anumolu L, Trujillo MF (2012) Evaluating the performance of the two-phase flow solver interFoam. *Computational Science & Discovery*, Madison, p 37