Numerical Investigation of Two-Phase Flow through a T-Junction Microchannel Reactor

Mohamed Mansour1,2, Akimaro Kawahara2 and Michio Sadatomi2
1. Department of Mechanical Power Engineering, Faculty of Engineering, Mansoura University, Mansoura 35516, Egypt
2. Department of Mechanical System Engineering, Kumamoto University, Kumamoto 860-8555, Japan

Received: July 07, 2013 / Accepted: July 09, 2013 / Published: January 20, 2014.

Abstract: Recently, microreactors have become available to be fabricated and used safely. The performance of these microreactors depends on the behavior of the multiphase flow hydrodynamics. Gas-liquid flow through T-junction microchannel reactor is simulated numerically using VOF (volume of fluid) method. 2-D (Two-dimensional) and 3-D (three-dimensional) models of the T-junction microchannel reactor were introduced to the simulations. Both 2-D and 3-D simulations for nitrogen-water flow were performed in the FLUENT (Fluent. Inc.) computational fluid dynamics package. The third direction effect has been studied by comparing the results of the 2-D and 3-D simulations with the published experimental data. Also, the bubble slug length was calculated for the 2-D and 3-D simulations. Furthermore, the hydrodynamics of the flow was studied for the 2-D and 3-D simulations, and compared with other experimental data. The pressure drop, mean bubble velocity, the velocity distribution and the void fraction were calculated and found to be in good agreement with published data.

Key words: Two-phase flow, VOF method, pressure drop, bubble slug length, void fraction.

1. Introduction

The growth in manufacturing techniques of microstructures leads to rapid increasing in microfluidic devices applications such as MEMSs (micro-electro-mechanical systems), integrated cooling of electronic circuits, microreactors, and liquid and gas chromatographs. Microdevices have had a major impact on many disciplines such as biology, medicine, optics, aerospace, and mechanical and electrical engineering. Microreactor is an example of the microstructure devices in which chemical reactions take place to produce chemical productions, hydrogen for use in fuel cells and biodiesel fuels (mixture of vegetable oil and solvents such as methanol and ethanol that is used as a diesel-equivalent fuel). Large surface area to volume ratio, small reaction volume and efficient mixing improve heat transfer, mass transport and reaction rate in microreactors [1, 2].

The T-junction microreactor is the most popular one due to its simplicity in construction and its wide applications. Multiphase flows are encountered in microstructured chemical reactors for gas-liquid and liquid-liquid reactions. Efforts have been done experimentally and theoretically to study the single-phase (gas and liquid) flow in microchannels to show that the flow in micro-scale channels are different from that in the macro-scale channels [3-9]. On the other side, the two-phase gas-liquid flow has been studied much less and previous studies suggested that the two-phase flow in microchannels is quite different from that in conventional channel. Thus, in order to design and fabricate micro-devices which deal with the two-phase flow effectively, such as microreactors and fuel cells, two-phase flow in the micro-scale needs more studies.

In the two-phase flow, the magnitude of forces acting on the different phases changes with the
channel size. For example, surface tension, viscous and inertia forces become much stronger as the channel size decreases while gravitational forces are weakened. Therefore, two-phase flow characteristics are more sensitive to dimensional scales than the single-phase flow [10, 11].

Previous studies on the two-phase flow has observed different flow patterns such as bubbly, slug, churn and annular flow regimes depending upon the flow properties and conditions. Suo and Griffith [12] studied experimentally the horizontal gas-liquid flow in circular microchannels. They used heptane and water as the liquid phase, and helium and nitrogen as the gas phase. Three flow patterns (slug, slug-bubbly and annular flow) were identified. On the other hand, they could not specify the stratified flow pattern. Also, Coleman and Garimella [13] studied experimentally the air-water flow in horizontal minichannels with circular and noncircular cross-sections to investigate the effect of tube diameter and shape on flow regime transitions. Bubble, dispersed, elongated bubble, slug, stratified, wavy, annular-wavy, and annular flow patterns were observed. Moreover, they developed flow regime maps and the transitions between the different flow regimes.

Yang and Shieh [14] investigated the flow patterns for air-water and refrigerant R-134a in horizontal tubes with inside diameter from 1.0 mm to 3.0 mm. Their experimental results agree well with literature for the air-water flow, however, for the R-134a flow, the flow patterns deviate from literature at low gas velocities. They concluded that the surface tension force is essential in determining the flow pattern maps. Air-water two-phase flow was studied in a near square microfluidic T-junction both experimentally and numerically by Santos and Kawaji [15]. Experimental results observed two different flow patterns (slug and stratified flow). Furthermore, the numerical gas slug length had the same size of the experimental gas slug length for low slug lengths (less than 400 μm) and had different lengths for high slug lengths (above 400 μm). They proposed that modifications were required to the numerical model in high shear rate flows.

Among all of these flow regimes, it is found that over a wide range of operating conditions, the flow in microchannels is slug flow (Taylor flow). Slug flow is characterized by uniform gas plugs filling the channel, separated by liquid plugs. The gas slugs are surrounded by a thin liquid film that connects the two successive liquid slugs separated by the gas bubble [16]. The gas slug length varies with the gas and liquid fluxes. Kawahara et al. [17-19] investigated the effects of liquid properties on two-phase flows characteristics by using different liquids experimentally. Distilled water and ethanol with different concentrations were used as the test liquids and nitrogen as the test gas. The two-phase flow through T-junction mixer microchannel was studied using two different mixers to investigate microchannel inlet effects. Two different flow patterns were observed; quasi-homogeneous flow at low superficial gas velocity and quasi-separated flow at relatively high superficial gas velocity. They measured gas slug length, void fraction and bubble velocity at different flow conditions.

Moreover, an isothermal two-dimensional flow through plane T-junction micro-channel with cross-sectional width of 0.5 mm was studied by Basurco and Nieckele [20]. Their results were compared with other numerical and experimental data and a good agreement was obtained. Pham et al. [21] studied the microbubble formation in a T-junction microchannel numerically. They analyzed the pressure distribution along the microchannel, the velocity distribution and the flow patterns. Furthermore, a numerical flow pattern map was obtained and compared with other experimental data. Santos and Kawaji [22] studied numerically and experimentally the effect of changing the gas and liquid fluxes on the air-water slug flow characteristics (slug length, slip velocity, void fraction and pressure drop).

Gas-liquid flow through T-junction microchannel with varying cross-sectional width (0.25, 0.5, 0.75, 1,
Numerical Investigation of Two-Phase Flow through a T-Junction Microchannel Reactor

2 and 3 mm) was studied numerically by Qian and Lawal [11]. They calculated gas and liquid slug lengths at various operating conditions for 2-D (two-dimensional) and 3-D (three-dimensional) flow to examine the third direction effect. There were little differences in the results between the 2-D and 3-D slug lengths. They proposed that these differences are due to low resolution of the 3-D simulations. So they ignored the effect of the third direction and used the 2-D simulations in their following simulations. On the other hand, Desai et al. [23] simulated 2-D and 3-D air-water flow in 0.5 mm circular T-junction microchannel. Their 2-D model failed to predict correctly the slug flow pattern. Also, Santos and Kawaji [15] failed to capture the real slug flow pattern at high shear rate for 2-D model. The slugs were unequal in sizes and shapes.

It is clear from the above review that many publications have studied two-phase flow in microchannels both experimentally and numerically. Most of the numerical studies performed 2-D simulations and ignored the third direction effect. In addition, there is a noticeable disagreement between scientists whether or not the third direction has any effect on flow patterns and characteristics (the gas and liquid slug lengths, and the pressure drop). Therefore, the aim of the present study is to construct 2-D and 3-D numerical models that are capable to capture the real behavior of the two-phase flow in a T-junction microchannel (based on that used by Kawahara et al. [17]) and compare between them to investigate the effect of the third direction. Also, results are compared with other experimental data [17].

2. Mathematical Model

2.1 Governing Equations

The mass, momentum and gas volume fraction \( \alpha_G \) conservation equations for an incompressible two-phase flow using VOF (volume of fluid) method can be written as:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0
\]

\[
\frac{\partial (\rho \vec{V})}{\partial t} + \nabla \cdot (\rho \vec{V} \vec{V}) = -\nabla p + \nabla \left[ \mu \left( \nabla \vec{V} + \nabla \vec{V}^T \right) \right] + \rho \vec{g} + \vec{F}
\]

and

\[
\frac{\partial (\alpha_G)}{\partial t} + \vec{V} \cdot \nabla \alpha_G = 0
\]

where, \( \rho, \mu, \vec{V}, p \) and \( \vec{F} \) denote the mixture density, the mixture viscosity, the mixture velocity field, the pressure and the source term that represents the surface tension force, respectively. \( \rho, \mu \) and \( \vec{F} \) can be computed as a function of liquid and gas volume fractions (\( \alpha_L \) and \( \alpha_G \)),

\[
\rho = \alpha_L \rho_L + \alpha_G \rho_G
\]

\[
\mu = \alpha_L \mu_L + \alpha_G \mu_G
\]

\[
\vec{F} = \frac{2 \rho G G \nabla \alpha_G}{(\rho_G + \rho_L)}
\]

\( K_G \) is the curvature computed from the divergence of the unit surface normal, and \( \sigma \) is the surface tension. The value of \( \alpha_L \) and \( \alpha_G \) should be between 0 and 1.

3. Model Geometry

An unsteady nitrogen-water (gas-liquid) flow was simulated through a 2-D and 3-D T-junction microchannel reactor to investigate the two-phase flow characteristics in the test section (reaction channel) and to study the third direction effect. The T-junction microchannel reactor consists of two mixing channels (vertical one for nitrogen inlet and horizontal one for water inlet) and one horizontal reaction channel as shown in Fig. 1 based on that studied experimentally by Kawahara et al. [17]. Each one of the two mixing channels has an internal diameter of 250 \( \mu \)m and a length of 750 \( \mu \)m while the reaction channel has an internal diameter of 250 \( \mu \)m and a length of 3 cm. In
the reaction channel, the ratio of \((L/d = 120)\) is enough to ignore the entrance and exit effect. Knowing that the length of the reaction channel is increased, the computational time will increase rapidly.

The geometry and mesh were generated in GAMBIT software (part of FLUENT package) using quadrilateral elements. A grid independence study was performed and it is found that the grids with 8,568 cells and 268,541 cells are fine enough and suitable for the calculations for the 2-D and 3-D models, respectively.

4. Numerical Approach

Firstly, the T-junction microchannel reactor was filled with water. Then, nitrogen and water were fed from the vertical and horizontal mixing channels, respectively. Two-phase flow is assumed to be unsteady and laminar. At the inlet, uniform entrance volume flux is applied (the water inlet volume flux \((J_L)\) is kept constant at 0.4 m/s while the nitrogen inlet volume flux \((J_G)\) varied between 0.1 m/s and 1 m/s), and at the exit, the gradients of the flow variables are kept at zero. At the wall, a no-slip boundary condition is imposed. Also, simulations were performed in the FLUENT (Fluent. Inc.) computational fluid dynamics package, which utilizes the finite volume method for the spatial discretization.

The VOF algorithm was implemented to model the gas-liquid flow through the T-junction microchannel reactor. VOF is a numerical technique which is used to track and locate the shape and position of the interface between two or more immiscible fluids. Moreover, the momentum equation Eq. (2) along with the gas volume fraction \((a_G)\) conservation equation Eq. (3) are used to calculate \(a_G\) throughout the domain. The different phases can be identified by knowing the value of \(a_G\) (1 for gas phase, 0 for liquid phase and \(0 < a_G < 1\) for the interfaces). The interface is advected with the flow field through the numerical domain [16].

For all test cases, the interpolations for velocities and pressure are based on second order. The pressure-velocity coupling is obtained using the PISOs (pressure-implicit with splitting of operators) scheme. The PRESTO (pressure staggering option) and the geometric reconstruction schemes were used for the pressure and interface interpolation, respectively.

A segregated time dependent unsteady solver was used. Courant number was fixed at 0.25 and time step ranged between \(1 \times 10^{-5}\) s and \(1 \times 10^{-7}\) s depending on the inlet nitrogen volume flux to achieve the convergence of the solution. The surface tension is set to 0.072 N/m and wall contact angle of water is set to 0°. The density of water and nitrogen is 998.2 kg/m³ and 1.138 kg/m³, respectively.

5. Results and Discussion

The hydrodynamic behavior of nitrogen-water flow through T-junction microchannel is investigated numerically. The 2-D simulations were performed on the proposed T-junction microchannel at a fixed inlet water volume flux \((J_L = 0.4\) m/s) and varied inlet nitrogen volume fluxes ranging from 0.1 m/s to 1 m/s. Fig. 2 shows contour plots of volume fraction of Nitrogen in a 250 μm microchannel at different inlet
Numerical Investigation of Two-Phase Flow through a T-Junction Microchannel Reactor

Nitrogen volume fluxes. Two different flow patterns are obtained.

The first one (Fig. 2a) is the slug (quasi-homogeneous) flow pattern which characterizes by the formation of relatively small plugs of similar sizes and this flow pattern occurs at relatively small gas flow rate. The second one (Fig. 2b) is a stretched slug flow pattern which features by formation of very long slugs with irregular shapes and sizes. This flow pattern occurs at relatively high gas flow rates. The same simulations were repeated for the 3-D geometry model at the same inlets and outlet boundary conditions to study the third dimension effect. The slug flow patterns appeared at the relatively low gas velocity as in the 2-D simulations. While at the relatively high gas inlet volume flux, longer gas bubbles regular shape and size. These bubbles are different from the stretched slug flow patterns which appear in the 2-D simulations as shown in Fig. 3. The bubble slug length increases with the increasing of the inlet gas volume flux.

Moreover, the results of the simulations which were performed with 2-D and 3-D geometries were compared with the experimental results obtained by Kawahara et al. [17] (Fig. 4). The comparison shows that the 2-D simulations fail to capture the real flow patterns of the two-phase flow in microchannels at relatively high gas inlet volume flux while it gives satisfactory results at relatively low gas inlet volume flux.

On the other hand, the 3-D simulations give satisfactory results at both low and high gas inlet volume flux. Also, the gas slug length were computed at different gas volume fluxes for 2-D and 3-D simulations and compared with the experimental results of Kawahara et al. [17] as shown in Fig. 5.

The 3-D simulation results agree well with the experimental data (the maximum difference reaches to 5%, while it reaches to about 39% for the 2-D simulations). The bubble slug length for 2-D simulations were computed only at small gas volume fluxes because at higher gas volume fluxes, the bubbles become irregular in shapes and sizes, and their bubble lengths are meaningless.

The above discussion explains the disagreement between Qian and Lawal [11] and Desai et al. [23] about the effect of the third direction. The deviation between their results may be due to the boundary conditions and the surface tension force values.

The 3-D simulations require too long time and large computer capacities compared to that for the 2-D simulations. Therefore, it is more economical to determine the limits at which the 2-D simulations can be used. Therefore, more 2-D simulations were performed at different homogenous volume fractions ($\beta = J_G / (J_G + J_L)$) ranging from 0.2 to 0.714 as shown in Fig. 6. As $\beta$ increases, the bubble slug length increases and the number of slugs decreases. The bubbles are regular in shape and size at low values of $\beta$ (less than 0.3043). The irregularity appears slightly at $\beta$ equals to 0.316 ($J_G = 0.185$ m/s and $J_L = 0.4$ m/s) and increases with increasing of $J_G$ and the number of slugs decreases. At $\beta = 0.714$ ($J_G = 1$ m/s and $J_L = 0.4$ m/s), there are only two irregular slugs. Therefore, below $\beta = 0.3043$ ($J_G = 0.175$ m/s and $J_L = 0.4$ m/s), 2-D geometries can be simulated with acceptable magnitude of error. 2-D and 3-D simulation were compared in Figs. 7 and 8 at $\beta = 0.333$ ($J_G = 0.2$ m/s and $J_L = 0.4$ m/s) and $\beta = 0.6$ ($J_G = 0.6$ m/s and $J_L = 0.4$ m/s), respectively. As discussed before, in the 3-D simulations, regular shape and size slugs are appeared rather than that in the 2-D simulations.

Pressure distribution along the test section axis is computed for 2-D and 3-D simulations as shown in Fig. 9. The upper lines represent the pressure distribution through the gas slugs while the below ones are the pressure distribution through the liquid slugs. The pressure distribution of the 3-D simulations is higher than that of the 2-D simulations because the bubbles are spheres in the 3-D simulations and both radii of curvature are taken into account in the calculation [11]. The pressure drop along the axial
direction consists of two components. The first one is the Laplace pressure drop which is the pressure difference between any two following gas and liquid slugs and it results from the surface tension effects [11]. The second component is the friction pressure drop which is the pressure difference between the front and back interfaces of any gas or liquid slug and it results from the viscous effects. The
Numerical Investigation of Two-Phase Flow through a T-Junction Microchannel Reactor

(a) $\beta = 0.2$ ($J_G = 0.1$ m/s and $J_L = 0.4$ m/s) and $t = 2.59$ s

(b) $\beta = 0.286$ ($J_G = 0.16$ m/s and $J_L = 0.4$ m/s) and $t = 0.098$ s

(c) $\beta = 0.298$ ($J_G = 0.17$ m/s and $J_L = 0.4$ m/s) and $t = 0.0549$ s

(d) $\beta = 0.3043$ ($J_G = 0.175$ m/s and $J_L = 0.4$ m/s) and $t = 0.126$ s

(e) $\beta = 0.316$ ($J_G = 0.185$ m/s and $J_L = 0.4$ m/s) and $t = 0.0591$ s

(f) $\beta = 0.322$ ($J_G = 0.19$ m/s and $J_L = 0.4$ m/s) and $t = 0.054$ s

(g) $\beta = 0.333$ ($J_G = 0.2$ m/s and $J_L = 0.4$ m/s) and $t = 0.0666$ s

(h) $\beta = 0.394$ ($J_G = 0.26$ m/s and $J_L = 0.4$ m/s) and $t = 0.0546$ s

(i) $\beta = 0.5$ ($J_G = 0.4$ m/s and $J_L = 0.4$ m/s) and $t = 0.1138$ s

(j) $\beta = 0.6$ ($J_G = 0.6$ m/s and $J_L = 0.4$ m/s) and $t = 0.0304$ s

(k) $\beta = 0.714$ ($J_G = 1$ m/s, and $J_L = 0.4$ m/s) and $t = 0.0262$ s

Fig. 6 2-D contour plots of volume fraction of nitrogen at different inlet N$_2$ volume fluxes.

Fig. 7 Comparison between 2-D and 3-D contour plots of volume fraction of nitrogen at $\beta = 0.333$ ($J_G = 0.2$ m/s and $J_L = 0.4$ m/s) and $t = 0.0496$ s.

Fig. 8 Comparison between 2-D and 3-D contour plots of volume fraction of nitrogen at $\beta = 0.6$ ($J_G = 0.6$ m/s and $J_L = 0.4$ m/s) and $t = 0.0304$ s.
Numerical Investigation of Two-Phase Flow through a T-Junction Microchannel Reactor

![Graphs of pressure distribution along the microchannel axis at different times for 2-D and 3-D simulations.](image)

**Fig. 9** Pressure distribution along the microchannel axis at $\beta = 0.2$ ($J_G = 0.1$ m/s and $J_L = 0.4$ m/s) and $t = 0.0628$ s for (a) 2-D simulation and (b) 3-D simulation.

The two-phase pressure drop due to friction can be expressed as:

$$
\left( \frac{dP_f}{dz} \right)_{TP} = \phi_L^2 \left( \frac{dP_f}{dz} \right)_L
$$

where, $\left( \frac{dP_f}{dz} \right)_{TP}$, $\left( \frac{dP_f}{dz} \right)_L$ and $\phi_L^2$ are the two-phase pressure drop due to friction, liquid pressure drop (liquid flows alone in the same microchannel) and the two-phase multiplier, respectively.

The two-phase multipliers can be computed using the C-coefficient method as:

$$
\phi_L^2 = 1 + \frac{C}{X} + \frac{1}{X^2}
$$

where, $X$ is the Lockhart-Martinelli parameter, $C$ is a coefficient which is fixed at 5 by Chisholm [24] in the case of laminar-laminar flow, and $\phi_L^2$ is the ratio between liquid pressure drop due to friction (liquid flows alone in the microchannel) and gas pressure drop due to friction (gas flows alone in the microchannel). $C$ is a coefficient which is fixed at 5 by Chisholm [24] in the case of laminar-laminar flow.

Kawahara et al. [17] stated that the coefficient $C$ is a function of the geometry and flow properties. They used the following empirical correlation:

$$
C = 1.38 Bo^{0.04} Re_L^{0.25} We_G^{0.12}
$$

where, $Bo$, $Re_L$ and $We_G$ are the Bond number, liquid Reynolds number and gas Weber number, respectively.

The two phase pressure drop due to friction calculated in Eq. (7) is compared with the obtained average Laplace pressure for the 2-D simulations and 3-D at $\beta = 0.2$ ($J_G = 0.1$ m/s and $J_L = 0.4$ m/s) are 550 Pa and 1,100 Pa, respectively. Qian and Lawal [11] concluded that the average Laplace pressure for the 3-D simulation is twice that of the 2-D simulations which agrees well with the obtained results.
The numerical results are in good agreement with the calculated \( \left( \frac{dP_f}{dz} \right)_{TP} \).

The numerical \( \left( \frac{dP_f}{dz} \right) \) is closer to \( \left( \frac{dP_f}{dz} \right)_{TP} \) than the experimental \( \left( \frac{dP_f}{dz} \right) \).

To study the effect of the third direction on the velocity distribution, the axial velocity distributions through a sectional line inside the gas bubble and liquid slug for 2-D and 3-D simulations were compared at \( \beta = 0.2 \) (because in this case the flow patterns of the 2-D simulations predict the flow patterns correctly) as shown in Figs. 11 and 12, respectively. The maximum axial velocity for the 3-D simulation is higher than that for the 2-D simulations (the difference reaches to 11.5% for the gas bubble and 3.11% for the liquid slug). The flows in both 2-D and 3-D simulations are asymmetry due to the mixing effect in the mixing junction.

Zuber and Findlay [25] used the drift flux model to develop the following relation for calculating the mean bubble velocity \( u_G \) through a horizontal duct:

\[
 u_G = C_0 J \tag{10}
\]

Where, \( u_G \), \( C_0 \) and \( J \) are the mean bubble velocity, the distribution parameter and the total volumetric flux \( (J = J_G + J_L) \), respectively. As discussed before in the C-coefficient method, Kawahara et al. [17] proposed that \( C_0 \) is a function of the geometry and flow properties. So they formulated a new relation for \( C_0 \):

\[
 C_0 = 3Bo^{0.10} Re_L^{-0.01} We_G^{0.01} \tag{11}
\]

The calculated mean bubble velocity are compared with the obtained 3-D numerical results and the experimental data of Kawahara et al. [17] as shown in Fig. 13. The solid line represents experimental data fitting line \( (u_G = 1.1J) \). The numerical bubble velocity was calculated for the 3-D simulations based on the mean velocity value through the gas bubble in the middle of the reaction microchannel. The numerical mean bubble velocity is closed to the experimental fitting line.

By way of example, the axial velocity distribution was calculated for the 3-D simulations at the middle of the microchannel through a sectional line inside the gas bubble at \( \beta = 0.714 \) \((J_G = 1 \text{ m/s} \text{ and } J_L = 0.4 \text{ m/s})\) as shown in Fig. 14. Furthermore, the velocity distribution through a sectional line inside the liquid slug was calculated as shown in Fig. 15. In comparison, the velocity profile is more blunted through the liquid slug and it increases with increasing of the gas inlet volume flux. Also, the maximum axial velocity is higher among the gas bubbles. The void fraction values for the 3-D simulations were calculated and compared with the experimental data of Kawahara et al. [17] at different values of \( \beta \) as shown in Fig. 16. The solid line in Fig. 16 represents the homogenous flow.
Fig. 11  Axial velocity comparison between 2-D and 3-D at $\beta = 0.2$ ($J_G = 0.1$ m/s and $J_L = 0.4$ m/s) through a sectional line inside the gas bubble.

Fig. 12  Axial velocity comparison between 2-D and 3-D at $\beta = 0.2$ ($J_G = 0.1$ m/s and $J_L = 0.4$ m/s) through a sectional line inside the liquid slug.

Fig. 13  Comparison between the mean calculated bubble velocity with obtained 3-D numerical results and experimental data of Kawahara et al. [17].
Fig. 14 Axial velocity distribution at $\beta = 0.714 \ (J_G = 1 \text{ m/s and } J_L = 0.4 \text{ m/s})$ through a sectional line inside the gas bubble.

Fig. 15 Axial velocity distribution at $\beta = 0.714 \ (J_G = 1 \text{ m/s and } J_L = 0.4 \text{ m/s})$ through a sectional line inside the liquid slug.

Fig. 16 Comparison between the 3-D numerical void fraction and the experimental data of Kawahara et al. [17].
line \( \alpha_G = \beta \), while the dashed line is the void fraction calculated by using Armand's correlation \[26\] \( \alpha_G = 0.833 \beta \). The numerical void fraction values are very close to the homogenous flowline. The numerical fitting line was found to obey the relation \( \alpha_G = 0.97 \beta \).

6. Conclusions

The effect of the third direction has been studied numerically for two-phase (nitrogen-water) flow through a T-junction microchannel reactor. 2-D and 3-D models were considered in these calculations. The results demonstrate that the flow patterns of the 3-D flow through the T-junction microchannel agree enough with the experimental data. On the other side, the flow patterns of the 2-D simulations give acceptable results only at lower values of \( \beta \). Furthermore, the following conclusions are drawn:

The pressure values of the gas bubbles are about twice that of the liquid slugs, which agrees with the other numerical results.

The 3-D pressure drop values agree well with other experimental data.

The bubble slug length increases with the increasing of the inlet bubble volume flux (with fixing the inlet liquid flux).

The maximum axial velocity for the 3-D simulation is always higher than that for the 2-D simulations.

The 3-D mean bubble velocity concurs well with the published experimental data.

The void fraction for the 3-D simulations are close to the homogenous flow line \( \alpha_G = \beta \).

References


Numerical Investigation of Two-Phase Flow through a T-Junction Microchannel Reactor


