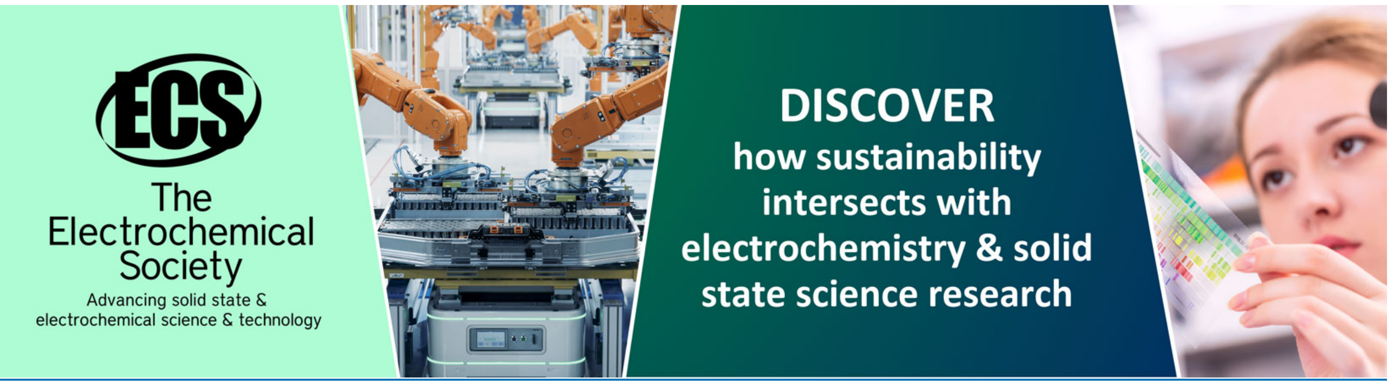
Journal of Physics: Conference Series

Modelling of two-phase flow in a minichannel using level-set method

To cite this article: H Grzybowski and R Mosdorf 2014 *J. Phys.: Conf. Ser.* **530** 012049



Journal of Physics: Conference Series **530** (2014) 012049 doi:10.1088/1742-6596/530/1/012049

Modelling of two-phase flow in a minichannel using level-set method

H. Grzybowski and R. Mosdorf
Department of Mechanics and Applied Computer Science, Faculty of Mechanical Engineering, Bialystok University of Technology
Wiejska 45C, 15-351 Bialystok, Poland

E-mail: h.grzybowski@doktoranci.pb.edu.pl

Abstract. Today there is a great interest in micro-scale multiphase fluid flow. In the paper, the numerical simulation of two-phase flow inside 3 mm minichannel was carried out. The liquid–gas interface was captured using the level-set method. During the calculation, the stabilization and reinitialization of level set function was performed in order to obtain the proper accuracy of the simulation. Incompressible Navier-Stokes equations were solved using the COMSOL Multiphysics® on a two-dimensional mesh. The process of formation of different two-phase flow patterns in the minichannel has been investigated. During the simulation it has been analysed three flow patterns: the bubbly flow and two kinds of slug flow with short and long slugs. It has been shown that unsteady flow at the inlet of the minichannel is responsible for the chaotic character of changes of the slug and bubble sizes. Such unsteady flow modifies the distance between the bubbles and slugs. It has been shown that for the low water inlet velocity the two-phase flow pattern becomes more stable.

1. Introduction

In recent years, the interest in simulation of two-phase flow in micro scale has significantly increased.The miniaturized channels are more frequently applied in devices such as heat exchangers, chemical reactors, and genome sequencers. Better understanding of two-phase flow phenomenon in minichannels induced the experimental and numerical investigations. Computational Fluid Dynamics gives the alternative approach which allows us to simulate two-phase flow in mini scale. Modelling of two-phase flow requires describing the interface behaviour. Over the years, different numerical techniques that dealt with fluid-interface motion were proposed. The two main groups of approaches for describing interface behaviour can be distinguished. Lagrangian technique uses markers to track the interface. In Eulerian techniques the interface is captured by a certain isocontour of a globally defined function. Eulerian techniques have the advantage over the Lagrangian techniques due to better adaptation to irregular contours, sharp corners, merging and breaking contours. Also, mixed Lagrangian-Eulerian methods were developed [1]. The most popular Eulerian techniques are: the level-set [2–4], volume-of-fluid (VOF) [5] and phase field methods [6, 7].
The level-set method was successfully applied to simulate gas flow in the liquid [8] – in these cases the contact of two phases has been considered. The main problem which occurs when we try to use the level-set method to simulate gas flow in the minichannel is the numerical stability of level set method in case of three phase contact. The main aim of the study is to simulate the two-phase flow inside the

minichannel using the level-set method. The CFD Module of COMSOL Multiphysics® has been used, the different flow conditions were analysed. The two-phase flow (water - air) was simulated inside the 2D minichannel with a length of 200 mm and height of 3 mm. The air was supplied to the minichannel by the small nozzle with height of 0.7 mm. The model parameters considered the three phases contact.

2. Level-set method

The level-set method, introduced by Osher and Sethian [3], allows to simulate flow of two immiscible fluids separated by moving interface. The level set function is typically smooth continuous function, denoted as ϕ . The interface is represented by 0.5 contour of the function ϕ . In the transition layer near the interface the function ϕ changes smoothly form 0 to 1. Regions where $\phi < 0.5$ are filled with water and region where $\phi > 0,5$ are filled with air.

$$\varphi \begin{cases} > 0{,}5 & air \\ 0{,}5 & interface \\ < 0{,}5 & water \end{cases} \tag{1}$$

The level set function is being advected with a flow field. The following equation describes the convection of the level set function [3]:

$$\frac{\partial \phi}{\partial \mathrm{t}} + \upsilon \cdot \nabla \phi = 0 \tag{2}$$

Main difficulty in numerical simulation of the moving boundaries is discontinuity appearing when fluid parameters changes at the interface. The problem was solved by defining fixed interface thickness where parameters could smoothly change. This procedure may cause mass conservation and constant interface thickness problems [9]. In CFD Module of COMSOL Multiphysics® the following level-set equation is solved [10]:

$$\frac{\partial \phi}{\partial \tau} + \upsilon \cdot \nabla \phi = \gamma \nabla \cdot \left(\varepsilon \nabla \phi - \phi(1-\phi)\frac{\nabla \phi}{|\nabla \phi|} \right) \tag{3}$$

This equations (3) have stabilization and reinitialization terms. The left side of the equation defines the motion of the interface while the right side is responsible for numerical stabilization and reinitialization. Parameter ε controls the interface thickness and parameter γ defines the intensity of reinitialization. In the model the density ρ and the dynamic viscosity μ are defined as follow:

$$\rho = \rho_1 + (\rho_2 - \rho_1)\phi \tag{4a}$$

$$\mu = \mu_1 + (\mu_2 - \mu_1)\phi \tag{4b}$$

where ρ_1 liquid density, ρ_2 air density, μ_1 and μ_2 are the dynamic viscosity of liquid and air respectively.
In the paper it have been solved the Naviera-Stokes equations:

$$\nabla \cdot \vec{u} = 0 \tag{5a}$$

$$\rho \frac{\partial \vec{u}}{\partial t} + \rho(\vec{u} \cdot \nabla)\vec{u} = \nabla \cdot \left[-pI + \mu\left(\nabla \vec{u} + \nabla \vec{u}^{\mathrm{T}}\right)\right] + \vec{F} + \rho \vec{g} + \sigma\kappa\delta\vec{n} \tag{5b}$$

where u is velocity, g is gravity, t is time, I identity matrix, p is pressure. Expression $\sigma\kappa\delta\vec{n}$ donates surface tension forces at the interface where σ is surface tension coefficient, κ is curvature, $\vec{n}$ is the unit normal to the interface and δ Dirac delta function concentrated to the interface. Curvature κ, interface normal $\vec{n}$ and Dirac delta δ are determined by [10]:

$$\vec{n} = \frac{\nabla\phi}{|\nabla\phi|},\ \kappa = \nabla(\frac{\nabla\phi}{|\nabla\phi|}),\ \delta = 6|\nabla\phi||\phi(1+\phi)| \tag{6}$$

3. Numerical implementation

The Navier-Stokes equations coupled with level set function are solved in COMSOL. The two-phase flow was simulated inside the minichannel with a length of 200 mm and height of 3 mm. The 2D domain under consideration has been shown in figure 1.

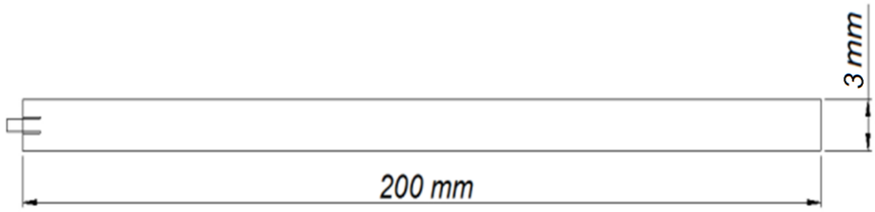


Figure 1. Computational domain

On left side of the channel the small nozzle supplying air was placed. The height of the nozzle was 0.7 mm. In figure 2 the schema of boundary conditions in the minichannel inlet has been shown.

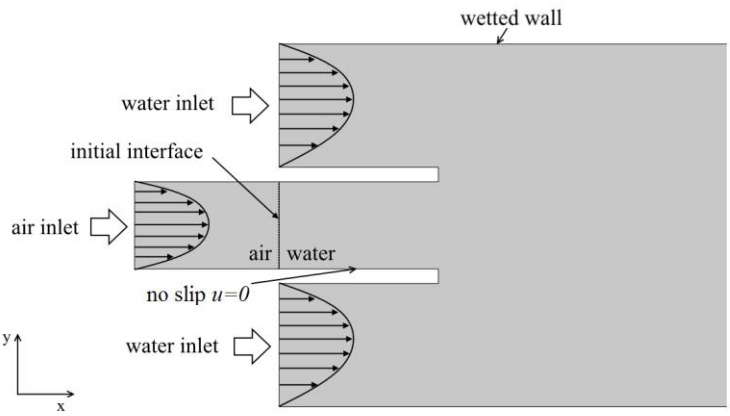


Figure 2. The schema of boundary conditions in the minichannel inlet.

Parabolic inlet velocity profile is given by the following equation:

$$u = u_{\max} s(1-s) \tag{7}$$

where $u_{\max}$ is maximum value of the velocity profile and s is normalized length of boundary.

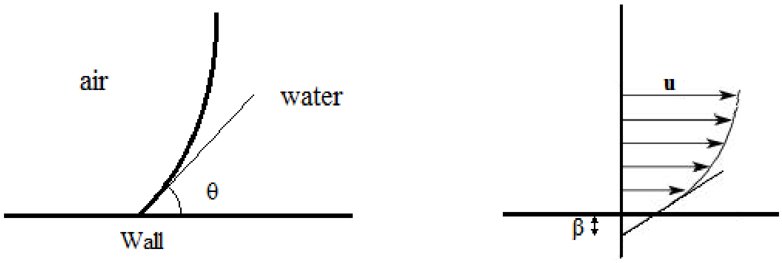


Figure 3: The definition of the contact angle θ at interface/wall contact points (left) and the slip length β (right).

The boundary conditions on the wall of the channel assume the contact angle θ (figure 3 a). In the simulation the value of contact angle was 20° (water – glass at 20^{o}C), the slip length (β) was equal to average mesh element size. In figure 3 the definition of contact angle and slip length has been shown. The mesh near the minichannel inlet is shown in figure.4.

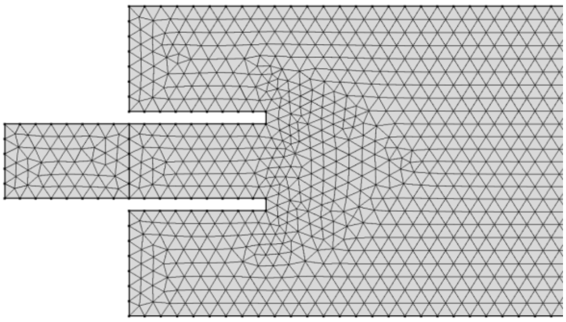


Figure 4: Mesh near the minichannel inlet.

In the simulation the parameter γ defining the intensity of reinitialization was 1 m/s and the parameter ε controlling interface thickness was equal to half of mesh size. The simulation was carried out for three different inlet conditions shown in table 1.

Table 1. Inlet velocity conditions.

No.	u_{max} water (ms^{-1})	u_{max} air (ms^{-1})
1	0.2	0.5
2	3	1
3	1	3

4. Simulation results

In the paper it has been analysed the process of formation of different two-phase flow patterns in the minichannel and the properties of two-phase flow patterns. During the simulation it has been obtained the three flow patterns: the bubbly flow and two kinds of slug flow with short and long slugs. The formation of bubbles and slugs in the inlet of the minichannel has been shown in figure 5. The flow patterns obtained at different inlet conditions have been shown in figure 6.

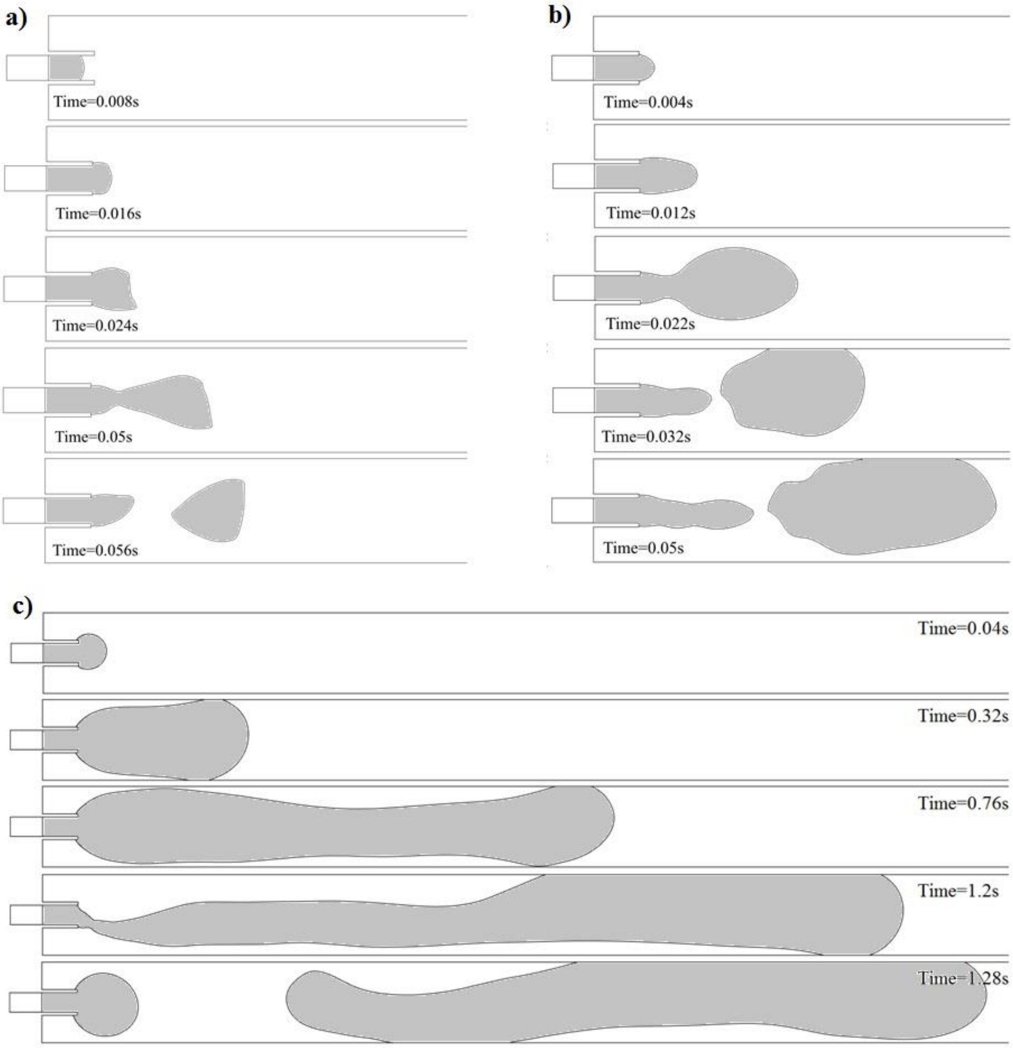


Figure 5: Bubble formation inside the minichannel inlet for different inlet conditions. a) water u_{max} = 3 ms^{-1}, air u_{max} = 1 ms^{-1}, b) water u_{max} = 1 ms^{-1}, air u_{max} = 3 ms^{-1} c) water u_{max} = 0.2 ms^{-1}, air u_{max} = 0.5 ms^{-1}

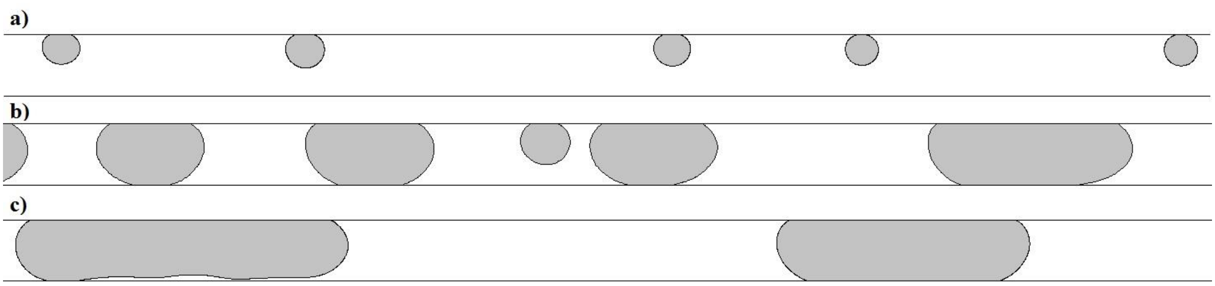


Figure 6: Two-phase flow for different inlet conditions at distance of 100 mm from the minichannel inlet: a) water u_{max} = 3 ms^{-1}, air u_{max} = 1 ms^{-1}, b) water u_{max} = 1 ms^{-1}, air u_{max} = 3 ms^{-1} c) water u_{max} = 0.2 ms^{-1}, air u_{max} = 0.5 ms^{-1}

For the high water inlet velocity (figure 5a) and the relatively low air inlet velocity in the minichannel the small bubbles are formed. The shapes of the bubbles show the existence of unsteady water flow condition in the minichannel inlet. At distance of 100 mm from the minichannel inlet the bubbles have a spherical shape and they move close to the upper wall of the minichannel (because of buoyancy force), figure 6a. In the case of lower water inlet velocity (figure 5b) and higher air inlet velocity the size of the bubbles increases and small slugs are formed in the minichannel inlet. At the distance of 100 mm from the minichannel inlet (figure 6b) the slug flow with short slugs and small bubbles has been observed. The shapes of the bubbles in the inlet of the minichannel indicate the existence of unsteady water flow conditions which causes that the size of subsequently formed bubbles changes during the simulations. The small water inlet velocity (figure 5c) causes the formation of the long slugs in the minichannel inlet. Such flow pattern is observed at the distance of 100 mm from the minichannel inlet (figure 6c). Unsteady character of liquid flow at the minichannel inlet causes that the lengths of subsequent slugs vary.
The bubbles and slugs join during their flow, (figure 6b). In figure 7 it has been shown the example of bubble and slug coalescence. In such a way the longer slugs are created.

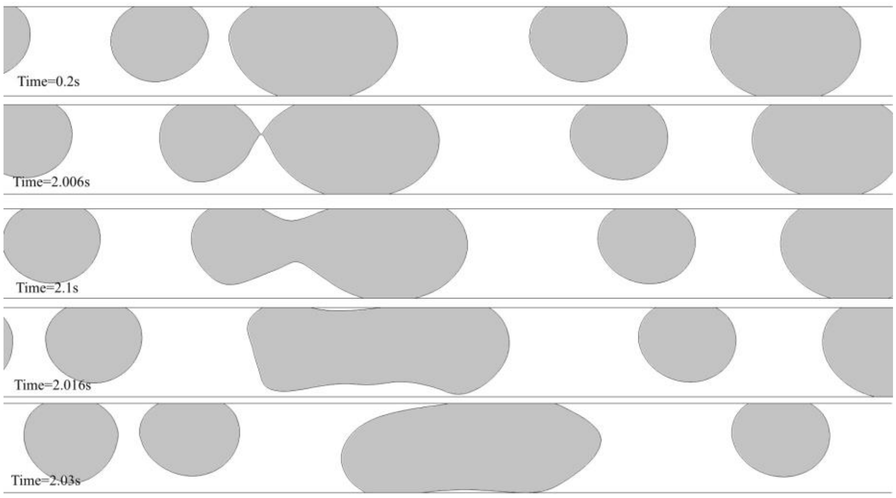


Figure 7: Bubble merging. (water u_{max} = 1 ms^{-1}, air u_{max} = 3 ms^{-1})

For evaluation of the dynamics of formation of two-phase flow patterns in the minichannel it has been used the ‘cut line’ method. The ‘cut line’ was located at the distance of 100 mm from the minichannel inlet. At the line the changes in time of void fraction (n) have been measured. In figure 8 it has been shown the recorded time series. The amplitude of functions $n(t)$ indicates the size of bubbles (figure 8a and b). The distance between maximum values of $n(t)$ corresponds to the distance between bubbles. When the amplitude of $n(t)$ is equal to one, then the slug flows through the minichannel at the distance of 100 mm from the inlet. The time period when the values of $n(t)$ are equal to 1 correspond to the time of passing the slug through the ‘cut line’.
The chaotic character of changes of the size of bubbles (figure 8a) indicates the existence of unsteady flow at the inlet of the minichannel. Such unsteady flow modifies hydrodynamic forces which are responsible for bubble departures from the nozzle. The different distance between the bubbles are visible in figure 8a. This unsteady flow is also responsible for the chaotic character of function $n(t)$ in case shown in figure 8b, where the short slugs and bubbles are visible. In case shown in figure 8c there are only the slugs, and the flow pattern seems to be more stable in comparison to the other cases.

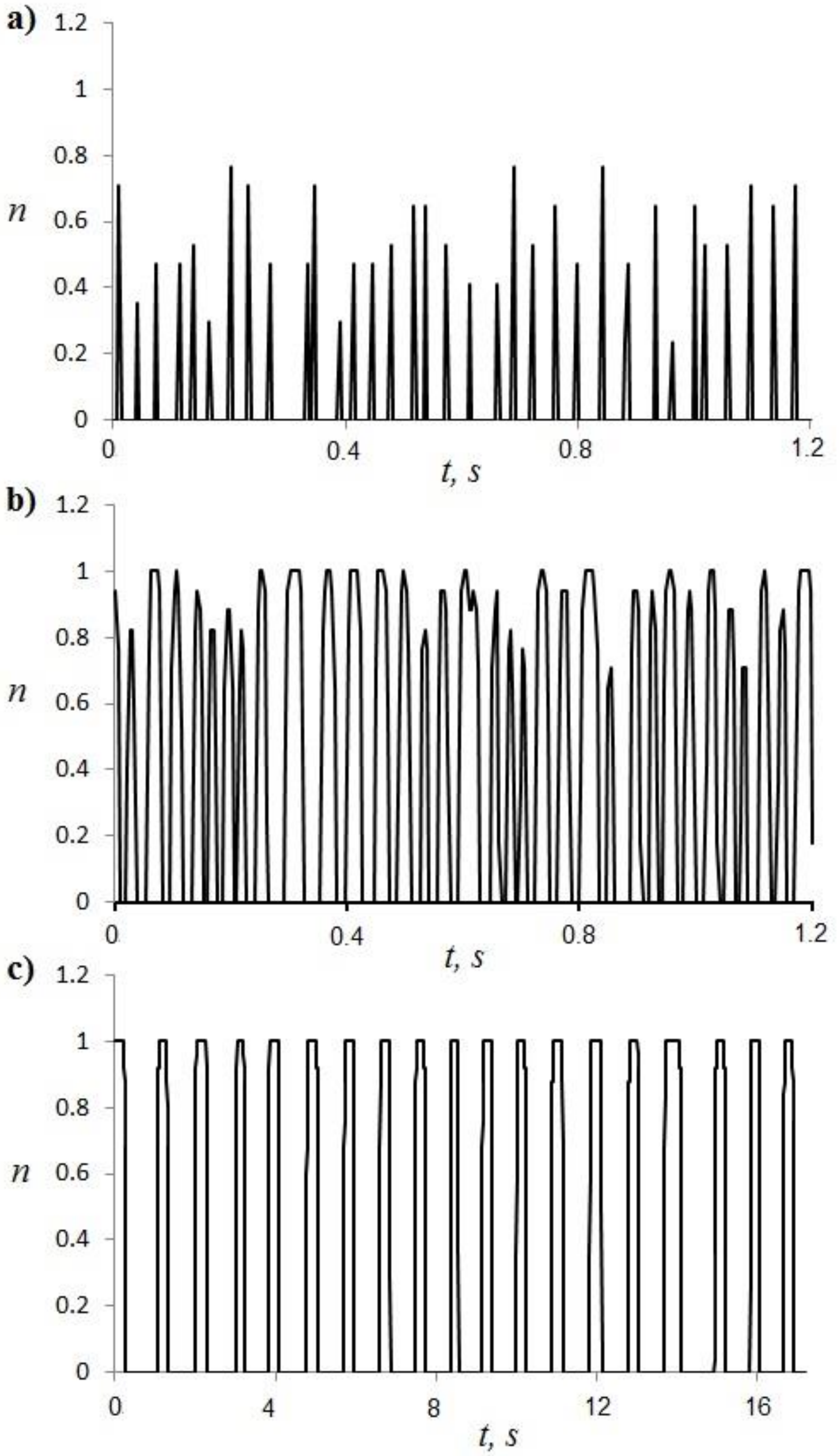


Figure 8: The dynamics of two-phase flow in minichannel. n is e number of mesh elements filed by air. a) water u_{max} = 3 ms^{-1}, air u_{max} = 1 ms^{-1}, b) water u_{max} = 1 ms^{-1}, air u_{max} = 3 ms^{-1} c) water u_{max} = 0.2 ms^{-1}, air u_{max} = 0.5 ms^{-1}

5. Conclusions

The main aim of the study was the simulation of two-phase flow inside the minichannel using the level-set method. The two-phase flow (water - air) was simulated inside the 2D minichannel with a length of 200 mm and height of 3 mm. The CFD Module of COMSOL Multiphysics® has been used. The air was supplied to the minichannel by the small nozzle. The simulation parameters considered the three phases contact.

The process of formation of different two-phase flow patterns in the minichannel has been investigated. During the simulation it has been analysed three flow patterns: the bubbly flow and two kinds of slug flow with short and long slugs. It has been shown that unsteady flow at the inlet of the minichannel is responsible for the chaotic character of changes of the slug and bubble sizes. Such unsteady flow modifies the distance between the bubbles and slugs. It has been shown that for the low water inlet velocity the two-phase flow pattern becomes more stable.

6. References

[1] Soulaimani A and Saad Y 1998 An arbitrary Lagrangian-Eulerian finite element method for solving three-dimensional free surface flows, *Comput. Method. Appl. M.* **162** (1–4) pp 79-106
[2] Olsson E and Kreiss G 2005 A conservative level set method for two phase flow. *J. Comput. Phys.* **210** 1 pp 225-246
[3] Osher S and Sethian J A 1988 Fronts propagating with curvature-dependent speed: algorithm based on Hamilton–Jacobi formations *J. Comput. Phys*. **79** pp 12
[4] Osher S and Fedkiw R 2002 Level Set Methods and Dynamic Implicit Surfaces, *Appl. Math. Sci.* **153** (Springer-Verlag)
[5] Rider W J and Kothe D B 1998 Reconstructing volume tracking *J. Comput. Phys.* **141** pp 112
[6] Chiu P and Lin Y 2011 A conservative phase field method for solving incompressible two-phase flows *J. Comput. Phys.* **230** (1) pp 185-204
[7] Sun Y and Beckermann C 2007 Sharp interface tracking using the phase-field equation *J. Comput. Phys* **220** pp. 626-653
[8] Mukherjee A, Kandlikar S G and Edel Z J 2011 Numerical study of bubble growth and wall heat transfer during flow boiling in a microchannel, *Int. J. Heat Mass Trans.* **54** (15–16) pp 3702-3718
[9] Ferziger J 2003 Interfacial transfer in Tryggvason's method, *Int. J. Numer. Meth. Fl.* **41** (5) pp 551–560
[10] COMSOL Multiphysics® User's Guide.

Acknowledgement
The project was funded by the Bialystok University of Technology - the number of decision: MB/WM/10/2013