TWO DIMENSIONAL SIMULATION OF MICROBUBBLE GENERATION IN A FLOW FOCUSING DEVICE

Ishwariya.R¹, Shankar N², Prathisha M³

¹,³Assistant Professor, Department of Medical Electronics Engineering, Saveetha Engineering College, Chennai, TamilNadu.

²Assistant Professor, Department of Biomedical Engineering, Saveetha Engineering College, Chennai, TamilNadu.

Email: ¹ishu4892@gmail.com, ²bmeshankar@gmail.com, ³prathisham7792@gmail.com

Abstract

Microbubbles have greatly increased its potential for medical applications such as medical imaging, drug and gene delivery. For numerical simulation, computational fluid dynamics is an important tool for analysing and developing a microfluidic system. We present a microfluidic co-flowing device which is able to produce a sequential pattern of monodispersed microbubbles. The two phase flow 2D minichannel was simulated using COMSOL multiphysics v4.3b software, where the central inlet for gas phase is located between two liquid phase inlets parallel to each other. During the simulation, interfacial flow mechanism occurred at the orifice and bubbly flow has been analysed from the outlet, due to the wettability between microbubble and the channel wall. The process of microbubble formation is highly sensitive to finite pressure range and its stability is dependent upon the low gas velocity. With this, produced microbubbles are uniform in size at a specified pressure range so that it is easy to predict the bubble diameter and its generation frequency.

Keywords: Microbubbles, Computational fluid dynamics, COMSOL, Interfacial flow, Orifice, Monodispersed microbubbles.

1. INTRODUCTION

Microbubbles of less than 10um has been used as contrast agents in medical imaging for many decades and now it has been found that it is also helpful in targeted drug and gene transfer in effect with ultrasound field. Since microfluidic technique is a promising tool in MEMS (Micro electro mechanical system)technology for microbubble generation, because it
offers high monodispersity and controllable size distribution. There are many research studies based on droplet production compared to bubble generation in a microfluidic system. Recently, different types of microfluidic device have been proposed for bubble generation using multiphase flow based on droplet flowing mechanism. Some experimental work suggested that, using eight flow focusing orifice in a parallel module causes dripping regime to produce uniform droplets at a flow rate of hundreds of kilohertz[4]. This method has a potential of producing massive number of droplets for biological use. Using flow focusing module, multilayer microbubbles have been produced by adjusting the flow rate ratio[5] and its size can also be varied. Multiple gas bubbles can also be produced in a co-focusing microfluidic channel when the pressure gradient between the gas liquid interface highly depends on the interfacial tension[6]. Bubble size will vary according to gas concentration and different flow rate ratio. Elamano pinto[7] carried out his work using flow focusing module to generate two forms of bubbles such as slug and spherical and studied about the flow of air microbubbles in vivo microvessels. Eleanor stride[8] and others investigated their work on monodisperse microbubbles using T-junction combined with electrohydrodynamic process. Comparing to experimental work results, simulation analysis is considerably low in bubble formation. Simulation work will provide various study of bubble formation in a microfluidic system and its calculation method in a theoretical manner. From the obtained simulation results it is easy to relate with experimental results and calculate its effectiveness.

In order to understand the flow dynamics and physics, Computational fluid dynamics (CFD) is a module that helps to solve multiphase flow phenomenon for a complex systems. Modelling of two phase flow needs interfacial behaviour which is responsible for periodic formation of microbubbles and during numerical simulation it gives the results of fluid-gas motion by using level set method. Generally, for a two phase flow analysis, various structure of microfluidic channel can be T-junction, Y-junction and Co-flowing device.

Some authors have published a detailed experimental and numerical study on gas-liquid Taylor slug flow in a microfluidic T-junction with nearly square microchannels of 100 μm nominal hydraulic diameter [1]. In this paper, they studied about the taylor slug formation in T-junction based on the principle of pressure drop caused by capillary forces at the gas-liquid interface. Due to pressure fluctuation, slug bubble flows from the interface in the channel, which shows the average velocity of gas is too high. And then, grzybowski [2] have observed three different types of bubble flow in a 2D minichannel with a length of 200 mm and height of 3 mm using level set method. They are in the form of slug flow, bubbly flow, long and short slugs mainly it happens due to the unsteady flow at the inlets of the
minichannel. This unsteady flow changes chaotic character of bubble size which modifies hydrodynamic forces responsible for bubble departure from the nozzle. So, pressure is not maintained at a controlled rate and by adding low water inlet velocity keeps unstable bubble generation. Formation of mini bubbles in a 2D T-junction was carried out by arias[3] using fluid dynamics numerical code JADIM. Here bubbles are generated as a result of capillary forces involved over inertia and buoyancy at the interface. After some period the tiny bubbles get deformed due to the gravitational force displacement from the capillary centre line. In this work, they obtain the results such as bubble generation frequency, void length, bubble length using gas and liquid flow rate with increased superficial velocity. Bubble size is unpredictable when it has different gas flow rate so that finally liquid phase will drag towards longitudinally in T-junction. Using flow focusing module, numerical simulation of microbubble analysis have not been reported much.

2. PROPOSED SYSTEM

2.1 Bubble breakup mechanism

In this paper, we propose a co-focusing model and it has two phases (continuous phase and dispersed phase) to flow parallel to each other. For this model dripping and jetting mechanism at the orifice is the main aspect of bubble formation. In the dripping process the breakup occurs right at the tip of continuous phase, while in the jetting process the bubbles pinch off from a jet (i.e. breakup of the bubble from the nozzle to the outer channel containing dispersed phase). The dripping occurs at low flow rates for both fluids while jetting occurs at high flow rates, of either the dispersed or the continuous phase. So here using pressure controlled mechanism, bubble breakup occurs within the orifice where the two phases are squeezed together by reducing the flow resistance in the model. Hence for this device, pressure controlled flow is given as main consequence in sequential production of bubbles compared to flow rate controlled mechanism.

2.2 Model structure

Simulation was performed in 2D using flow focusing device which has the geometry and dimension presented in Figure 1. This geometry has a small capillary within a square or rectangular outer channel, where the dispersed phase flows in the capillary and the continuous phase flows in the outer channel. It has three inlets where the dispersed phase (oil) is injected into the vertical inlet, while the continuous phase (gas) is introduced into the horizontal inlet. \( H_o \) and \( W_o \) represent the height and width of the outlet channel whereas \( H_i \)
and $W_i$ represent the height and width of inlet channel and $H$ represent height of the nozzle and $W$ represent width of the nozzle are $H_o=4\, \mu m$, $W_o=15\, \mu m$, $H_i=2\, \mu m$, $W_i=6\, \mu m$, $H=2\, \mu m$ and $W=1\, \mu m$. The contact angle is set to 0.349 rad.

![Diagram of the co-flowing device](image)

**Figure 1** Structure of co-flowing device

### 2.3 Materials and methods

A theoretical way to simulate the bubble formation by using time-dependent computations in COMSOL Multiphysics® is very much benefit for microfluidics domain. It is a simulation software that utilizes finite element analysis to solve various physics and engineering phenomena. Fluid flow is a module which were utilised for this project. The finite element analysis is a numerical method to solve partial differential equations. It has microfluidics and CFD module, for this work CFD offers wide range of options and it can different flow conditions. COMSOL 4.3b version is used for my work because it has so many features for fluid dynamics and it increases computation speed.

The simulations were based on the Navier-Stokes equations for laminar two-phase flow, level set (tpf). The two phases were set to ethanol vapour and fc40, with the necessary fluidic properties (density and dynamic viscosity) presented in table 1. For both inlet phases, the velocity field were often set as slightly different value while for outlet no viscous stress pressure boundary condition was used. Wetted wall is chosen as boundary condition for interaction of dispersed phase to the walls. A finer mesh with more elements provides a more accurate solution, here free tetrahedral mesh with a COMSOL Multiphysics® predetermined
element size of “extra fine” was utilized. Finally, finite element method utilizes an iterative process to calculate a solution to all couple field variables.

TABLE 1 Material properties of inlet phase

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>DENSITY</th>
<th>DYNAMIC VISCOSITY</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fc40</td>
<td>1865 Kg/m³</td>
<td>0.0041 Pa.s</td>
</tr>
<tr>
<td>Ethanol vapour</td>
<td>rho(pA[1/Pa],T[1/K])[kg/m³³]</td>
<td>eta(T[1/K])[Pa*s]</td>
</tr>
</tbody>
</table>

3. RESULTS AND DISCUSSION

We have examined this work using pressure controlled mechanism. Our results suggest that bubble size and shape is highly sensitive to pressure so that two-phase flow is possible only in a finite pressure window. Uniform size of bubbles is controlled by changing the velocity of the liquid phase. With the help of theoretical model it is easy to predict the equivalent diameter of themicrobubbles.

Figure 2 Volume fraction of fluid1 at T=0.04
Figure 2 shows the volume fraction of dispersed phase uses the volume of fluid (VOF) method to track and locate the fluid-gas interface where the bubbles drip off at the orifice so they are consistently producing large numbers simultaneously with uniform size. Finally, a series of bubbles is generated from $T=0$ to 0.08 and stability was maintained effectively throughout the outer channel. Figure 3 Shows the spherical bubble formation at $T=0.05$ reveals the dripping regime of bubble formation in a shear based system and maintained the pressure at a constant rate.

![Figure 3 Volume fraction of fluid1 at T=0.05](image)

![Figure 4 Velocity magnitude of fluid-gas](image)
In figure 4, it shows the study of effect of the velocity ratio between the two phases on bubble formation and this is responsible for squeezing regime for the breakup mechanism. If the velocity of any two phases gradually increases, bubble shape will get to deform. At the same time when analyzing the pressure in the model, the highest pressure occurs prior to the fluid entering the nozzle. At the narrowest point of the nozzle, the largest pressure drop occurs. This is a result of the pressure gradient as fluid flows from a high pressure region, fluid inlets, to a low pressure region. As the pressure kept at 300 and normal inflow velocity is given as 0.06 for gas phase whereas 0.09 is for liquid phase shows the continuous bubbly flow pattern without any transition in the breakup regime.

4. CONCLUSION

Simulations are a very convenient way to test new designs or approaches before the practical work is done. In this study, monodisperse microbubbles were successfully produced in a flow focusing model using CFD module in COMSOL software. With CFD, information about local parameters such as velocity and pressure fields, volumetric mass fractions and interface configurations are easily obtained, whereas in experimental work such data are difficult to measure. Our main aim of this study is to simulate the model using pressure controlled mechanism instead of flow rate controlled one using these parameters. So this simple mechanism reduces the flow resistance of the dispersed phase and it expands the pressure at the nozzle to capture the main features of bubble generation. Bubbly flow pattern is analysed when relative flow rate of the two fluids is highly sensitive to pressure and its stability depends on the normal inflow velocity for gas phase. By yielding good results from the simulation, this device will reproduce high bubble formation frequency with high stability in experimental work which is mainly useful for clinical use.

5. REFERENCES


**AUTHORS BIBLIOGRAPHY**

**Ms. R. Ishwariya** had her graduation in the year 2013 in Electronics and Communication Engineering from Anna University, Master of Engineering in (Medical Electronics) in the year 2016 from Anna University. Her research interests are in the area of Bio-MEMS. She has participated in many National Level and International level Conferences, Technical Symposium, workshops and seminars. She is currently working as an Assistant professor in Department of Medical Electronics, Saveetha Engineering College, Chennai.

**N SHANKAR** is an active researcher, currently pursuing his research in Osteoporosis (image Processing). He did his master’s degree (M.tech – Biomedical Instrumentation) at SRM University, bachelor’s degree Biomedical Engineering at Pondicherry University, 2011 and 2009 respectively. He has over 7.9 years of experience in teaching. He has published many papers in various International Journals, International Conferences, National Journals and National Conferences. His research interests are Image processing, Signal Processing, Biomedical Instrumentation, and Biometrics. He is currently working as an Assistant Professor in Department of Biomedical Engineering, Saveetha Engineering College, Thandalam, Chennai.
Ms. M. Prathisha had her graduation in the year 2013 in Electronics and Communication Engineering from Anna University, Master of Engineering in (Medical Electronics) in the year 2016 from Anna University. Her research interests are in the area of Bio-MEMS and Biomedical Instrumentation. She has published her papers in journals and has participated in many National Level and International level Conferences, Technical Symposium, workshops and seminars and Faculty Development Programs. She is currently working as an Assistant professor in Department of Medical Electronics, Saveetha Engineering College, Chennai. Prior to her current position.