TECHNICAL TRANSACTIONS | CZASOPISMO TECHNICZNE
MECHANICS | MECHANIKA
1-M/2016

LUTZ BÖHM, MARKUS KOLANO, MATTHIAS KRAUME*

SIMULATION OF THE SINGLE BUBBLE ASCENT WITH OPENFOAM

SYMULACJA WZNOSZENIA POJEDYNCZEGO PĘCHERZA ZA POMOCĄ PROGRAMU OPENFOAM

Abstract

This project is motivated by the challenge of cleaning flat sheet membrane surfaces with the help of aeration. On the basis of earlier experiments and CFD simulations, a decision was made to use the CFD-tool OpenFOAM in contrast to earlier simulations which were performed with Ansys Fluent. In the new simulations, the advancing computing power allowed the simulation of a bubble ascent in the full channel which is of special interest in cases where the bubble size is smaller than the channel depths. Besides saving the licensing cost, OpenFOAM allows access to the source code and, therefore, easier implementation of sub-models if necessary.

Keywords: computational fluid dynamics, OpenFOAM, bubble

Streszczenie

W pracy podjęto próbę poprawy możliwości czyszczenia płaskich powierzchni membrany za pomocą napowietrzania. Na podstawie wcześniejszych eksperymentów (z wykorzystaniem ANSYS Fluent) podjęto decyzję wykorzystania programu OpenFOAM – jednego z narzędzi CFD. W nowych symulacjach wzrost mocy obliczeniowej umożliwił symulację wznoszenia pęcherzyka w pełnym kanale. Oprócz oszczędności kosztów licencji, OpenFOAM umożliwia dostęp do kodów źródłowych, a więc łatwiejsze wdrażanie modeli podrzędnych.

Słowa kluczowe: CFD, OpenFOAM, pęcherz

DOI:

* DSc. Eng. Lutz Böhm, MSc. Eng. Markus Kolano, Prof. PhD. DSc. Eng. Matthias Kraume, Chair of Chemical and Process Engineering, Faculty of Process Sciences, Technical University of Berlin.

1. Introduction

The investigation of the ascent of a bubble in a quiescent liquid is a common topic in literature. It has been extensively discussed for Newtonian [1] and non-Newtonian liquids [2] as the continuous phase. In the frame of this project, the focus was specifically on cases motivated by the process of membrane filtration in flat-sheet membrane modules. In this case, narrow channels between the flat-sheet membranes with a rectangular cross-section (width >> depth; here: width: 160 mm and depth: 3÷7 mm; height: 1500 mm) which are filled with activated sludge (if used for waste water treatment) can be found. Due to the filtration process, a layer of deposits develops over time on the surface of the membrane. The thickness of the layer of deposits, which is proportional to the resistance of the filtration, can be controlled by inducing flows in the channel. One common method to induce this flow is aeration. Therefore, breaking it down to the basics, a bubbly flow in a channel with a rectangular cross-section with a non-Newtonian, co-current flowing continuous phase (activated sludge) is apparent in this process.

The aeration which controls the growth of the layer of deposits is doing so by inducing shear on the membrane surface through the liquid. No model is yet known that more or less accurately correlates the aeration rate and the induced shear or reduction of the layer of deposits This is also partly due to the fact that the even more simple system of one bubble rising in a narrow channel with a rectangular cross-section has not yet been sufficiently investigated. Based on this lack of knowledge, in the recent past, this project experiment-tally investigated this basic system [3]. The following parameters were varied: channel depth ($d_{c,}$ 3÷7 mm); bubble size (d_B, 3÷9 mm equivalent bubble diameter); superimposed liquid velocity (v_L, 0÷0.2 m/s); rheology of the continuous phase (Newtonian, non-Newtonian). High speed camera imaging, particle image velocimetry and electro-diffusion methods were applied as measurement techniques. This allows the analysis of the bubble behaviour [4, 5], the effect of the bubble on the surrounding liquid [6], and the local wall shear stress resulting from the liquid flow induced by the bubble ascent [7]. The last mentioned property, the local wall shear stress, is of especially high interest. Unfortunately, it is very demanding to actually measure this property. Only a few wall shear stress measurement techniques are available, but all of them have their shortcomings. Therefore, a CFD approach was chosen here to get a deeper insight into the influence of the ascent of a bubble on the resulting local wall shear stress. As is partially discussed in Prieske et al. [8], the idea of a numerical approach for this specific system is something that has already been realised by using Ansys Fluent. Mainly due to simulation capacity limitations, whilst this model was a good first approach, it also had its shortcomings. The most significant drawback was that only half of the channel depth was implemented in the grid used for the simulation. The measure to halve the cell number and, therefore, reducing the calculation time was chosen as no enhanced movement of the bubble was expected. Here, it has to be kept in mind that in most of the investigated cases, the bubble was squeezed into the channel as its equivalent bubble diameter was equal to or larger than the channel depth. Especially for cases with a bubble size smaller than the channel depth, this did not apply and, therefore, bubble movement normal to walls would be possible but is suppressed by the model.

These simulations were all performed in water. Basically, it would be possible to calculate a non-Newtonian case, as well. This would have increased the calculation demand significantly. Additionally, in the case of this commercial CFD tool, the implementation of this model is, up to a degree, a black box.

Based on the evolution of this project and increasing computational power in general, a new approach has been formulated which, at this point, has a large experimental database in the background for the purposes of validation and is able to overcome the shortcomings of prior simulations, including simulations from other groups in this field. The requirements for the new simulations were:

- transient, full 3D model to resolve the complex three-dimensional deformation and movement of the bubble during its ascent;
- sufficient resolution of the rising event, especially near the wall, as the wall shear stress induced by the bubble is of particular interest;
- adequate rising length/duration to reach the steady or stable periodic movement (from experience: more than one second of rising time)
- implementation of suitable non-Newtonian characteristic describing the behaviour of activated sludge after validation of the case with a Newtonian continuous phase.

In this project, OpenFOAM was used for the CFD simulations. As a free, open source program, it allows access to and, if necessary, adjustments to the source code. There are basically, no limitations to cell numbers or multithreading as often the case for commercial programs in the past.

2. Materials and Methods

OpenFOAM 2.0 was used for the simulation of the single bubble ascent. As the surface of the bubble was supposed to be resolved, a volume of fluid method was used as the multiphase model, implemented here with the help of interFoam. Due to the fact that in most cases of interest, the bubble ascent is a stable periodic movement (here: bubble size d_B = 3, 5 and 7 mm), a transient calculation had to be conducted. The grid used for the calculation (according to the experimental setup described, for example, in [5]), was 200 by 160 and 3 mm, 5 mm or 7 mm in height, width and depth (only the results for d_c = 3 mm are shown here) with a total number of approx. 500,000 cells. As is known from experience, it takes roughly one second from the beginning of the ascent until periodic movement of the bubble, 200 mm would not be sufficient as potential rising path. To minimise the calculation time, another approach was chosen here. The bubble was initialised 40 mm from the top of the grid.

Once the bubble moved one cell layer upwards, the lowest cell layer of the grid was deleted and a new cell layer was added at the top of the grid. Hence, the bubble was again in the same position as it was initially, relative to the top of the grid.

The OpenFOAM results shown here were, consequently, found for one channel depth of d_c = 3 mm and three bubble sizes d_B = 3 mm, 5 mm and 7 mm calculated in water without superimposed liquid velocity. The post-processing was performed with a Matlab code extracting all data of interest, e.g. the centroid of the bubble. In all cases, a rising duration of two seconds was simulated. As mentioned earlier, it takes roughly one second to gain a stable movement – this part was ignored for the analysis. The OpenFOAM data is mainly compared to results published in [5] for the same bubble sizes but a channel depth of d_c = 5 mm and 7 mm as comparable experiments were not performed in smaller channels.

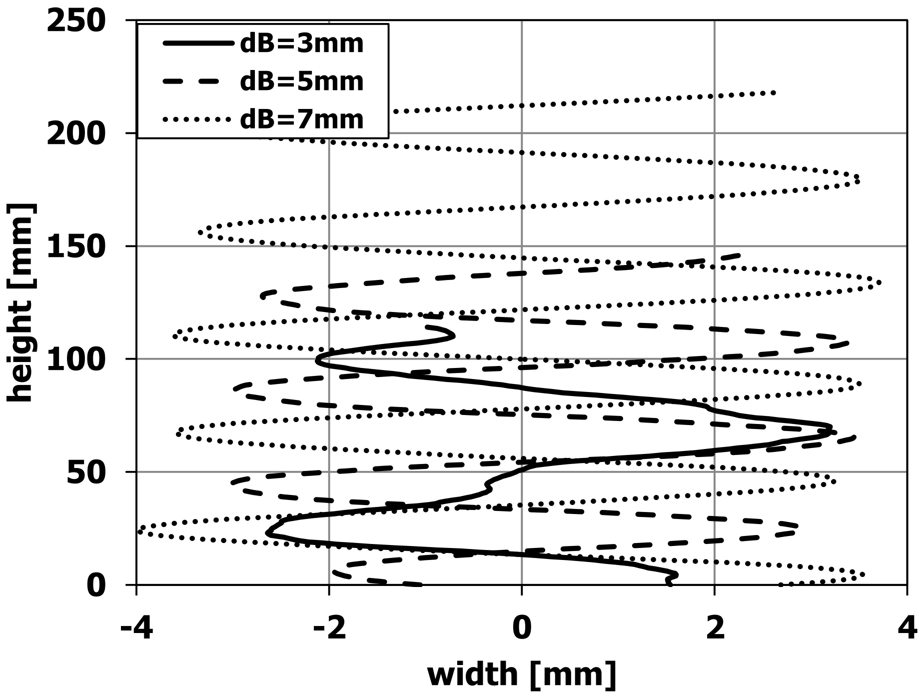

Fig. 1. Rising paths of three differently sized bubbles for a rising duration of 1 second each in a channel with a depth of d_c = 3 mm (from OpenFOAM simulation)

3. Results and Discussions

Figure 1 shows the rising path found with OpenFOAM for the three different bubble sizes. Illustrated is the position of the centroid with regard to both width and height. This illustration leads to three different ending points with regard to height for the three bubble sizes as all three of them have a different velocity.

While the movement of the smallest bubble seems slightly irregular, the other two bubble sizes show very smooth rising behaviour. Based on experience in confined geometries and, for example, Clift et al. [1] for free rising bubbles, a bubble with an equivalent diameter of d_B = 3 mm can perform several types of movements during its ascent, such as a straight, zigzag or helical movement. Due to the geometry, helical rising behaviour is not possible here. In the given case, this restriction leads to the zigzag movement. Qualitatively, the movements of all the bubble sizes look pretty comparable when it comes to, for example, the amplitude of the oscillation during its ascent. Based on experience with a larger channel depth [5], this is rather surprising.

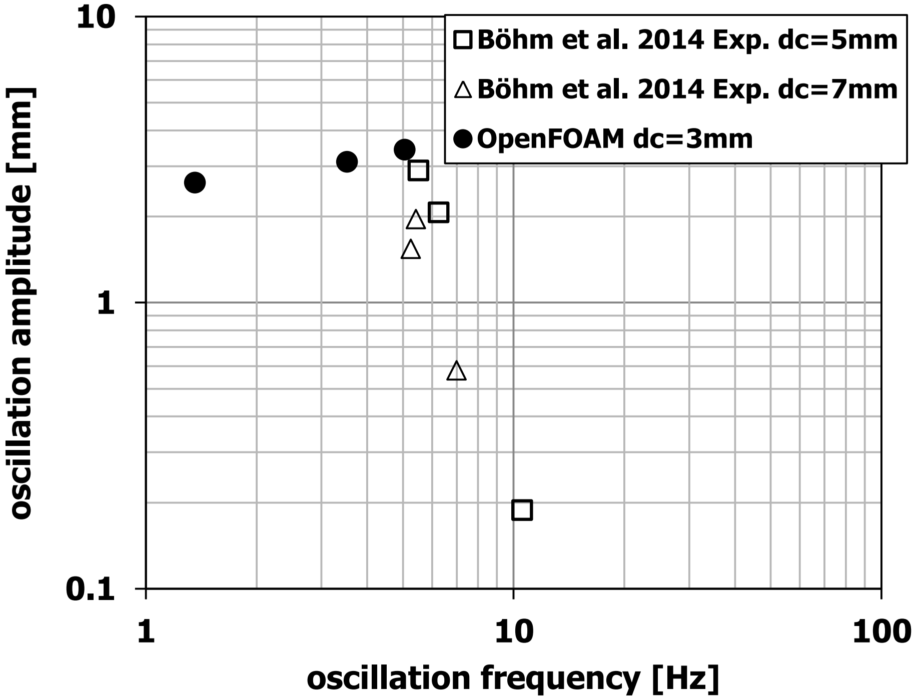

Fig. 2. Oscillation amplitude over oscillation frequency in a confining rectangular geometry with channel depths of d_c =5 mm and 7 mm (experimental data, [5]) and for a channel depth of d_c = 3 mm (OpenFOAM)

Figure 2 shows both, the quantitative results of the oscillation frequency and amplitude from the earlier experiments as well as the according results from the OpenFOAM simulation. The results found in the experiments in the larger channels tend to have a lower fluctuation in frequency and a broader range with regard to amplitude (with higher frequencies and lower amplitudes for smaller bubbles and vice versa for larger bubbles). On the other hand, in the simulation with the smaller channel depth, the results are contrary to those obtained with larger channels. The amplitudes are almost independent of the bubble size, but the frequency of the oscillation increases with the bubble size.

At this point, it is not clear if this differing behaviour is an effect of the channel depth or the simulation. One explanation could be that it is a result of the enhanced confinement in the 3 mm channel which leads to a stronger deformation of the bubble, which is the case for the smallest bubble size in particular. Figure 3 shows the ratio of the vertical and horizontal dimension of the bubble. Generally, the simulation and experimental results follow the trend that the ratio decreases with increasing bubble size, simply due to the enhanced confinement. Still, in case of the 3 mm bubble, the deformation is most enhanced for the smallest channel depth with a ratio below 0.8. This kind of deformation has an unavoidable effect on the bubble rising behaviour. Regarding the experimental data of the 3 mm bubble, it is worth mentioning, that due to experimental circumstances, the analysed high frequencies and low amplitudes mean that there is almost no oscillation at all. As mentioned earlier, in unconfined geometries, the 3 mm bubble is known to potentially behave in several ways. In the performed experiments, obviously straight movement appeared predominantly; however, from observations it can be stated that in some cases, the zigzag movement started in the analysed field of view or even earlier. This stochastic characteristic cannot be simulated properly, as the term ‘stochastic’ actually means in this case ‘under unknown conditions’. This means that there might be the possibility to set an initial condition for a simulation that leads to a straight movement of the bubble.

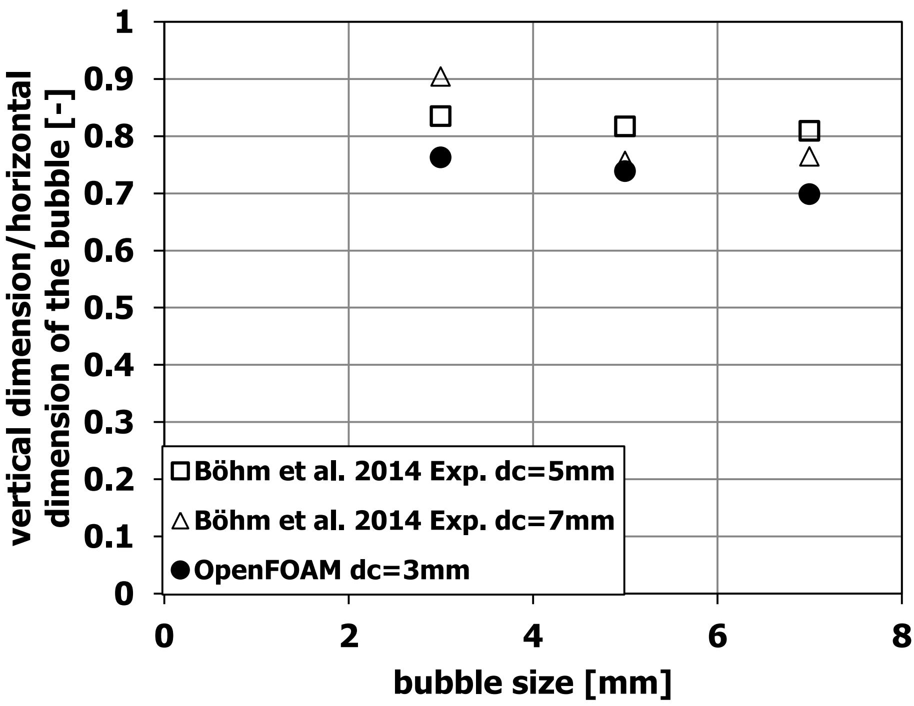

Fig. 3. Ratio of the horizontal and vertical dimension of the bubble over the bubble size in a confining rectangular geometry with channel depths of d_c = 5 mm and 7 mm (experimental data, [5]) and for a channel depth of d_c = 3 mm (OpenFOAM)

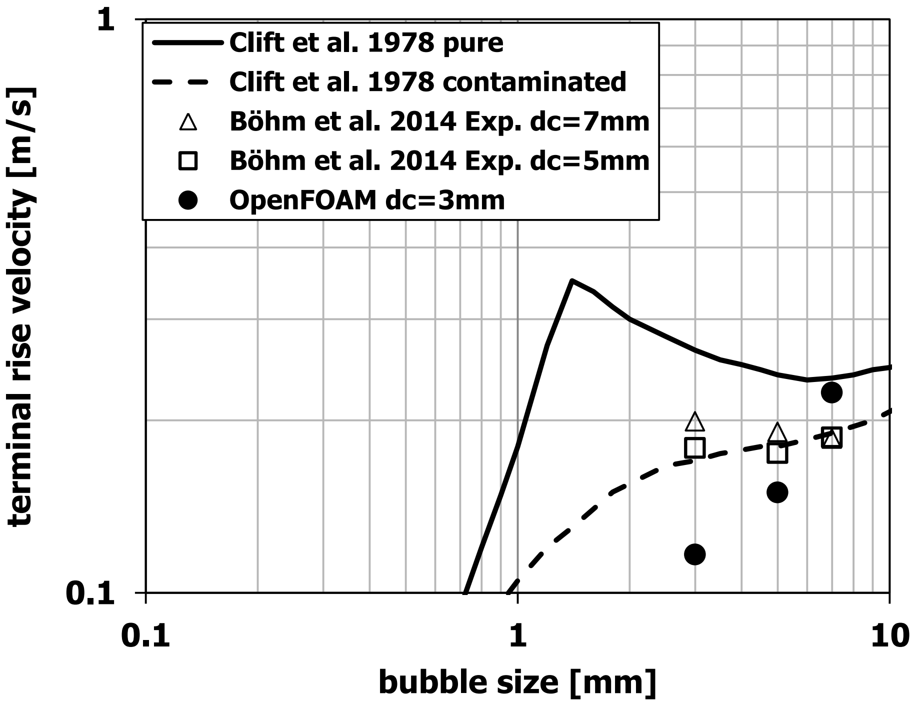

Fig. 4. Terminal rise velocity for free rising bubbles in pure and contaminated water (experimental data, [1]), in a confining rectangular geometry with channel depths of d_c = 5 mm and 7 mm (experimental data, [5]) and for a channel depth of d_c = 3 mm (OpenFOAM)

The effect of the deformation is also strongly visible in the trend of the terminal rise velocity of the bubble (Fig. 4). The experimental results are in agreement with the data found for free rising bubbles in contaminated water. Contamination results in a non-moving bubble surface which leads to a movement equal to that of a solid particle. Hence, although the experiments in [5] were performed in pure water, the confinement leads to a deceleration of the bubble. This behaviour is basically the same in the simulation results in the 3 mm channel while the deceleration is much more enhanced for the smaller bubbles. It is clear that a 3 mm bubble is much stronger influenced by the confining walls in a 3 mm channel than in a 5 mm or 7 mm channel. For the 3 mm and the 5 mm bubble, it is visible that the larger the channel depth, the higher the terminal rise velocity. Only in case of the 7mm bubble is the simulation value higher than in the experiments.

4. Conclusions

A simulation of the single bubble ascent in a confining rectangular geometry was performed with OpenFOAM. The developed model proved to be able to reproduce the oscillating movement during the bubble ascent, as it was also found in earlier experiments. The results shown here were simulated in a first channel depth that was not extensively tested in the experimental test phase. The several discussed effects on the oscillation parameters, the deformation and the terminal rise velocity cannot finally be attributed to be an effect of the enhanced confinement in the 3 mm channel (although potential explanations are given). These might also be an impulse for further model improvements. However, the developed model allows an extension to test cases with superimposed liquid velocities as well as cases with a non-Newtonian rheology of the continuous phase.

Acknowledegments

This work is part of the Collaborative Research Center/Transregio 63 "Integrated Chemical Processes in Liquid Multiphase Systems" (subproject A6). Financial support by the Deutsche Forschungsgemeinschaft (DFG, German Research Foundation) is gratefully acknowledged (TRR 63). Special thanks for the support during this work go to Phillip Rühl.

References

[1] Clift R.J., Grace J.M., Weber M., *Bubbles, drops and particles*, Academic Press New York 1978.

[2] Chhabra R., *Bubbles, drops, and particles in non-Newtonian fluids*, CRC Boca Raton 2006.

[3] Böhm L., *Comparison of single bubble and bubble swarm behavior in narrow gaps inside flat sheet membrane modules*, Technische Universität Berlin 2015.

[4] Böhm L. Kraume M., *Hydrodynamic investigation of single bubbles*, Czasopismo Techniczne Mechanika, vol. 109(5), 2012, 21-29.

[5] Böhm L.T. Kurita T. Kimura K., Kraume M., *Rising behaviour of single bubbles in narrow rectangular channels in Newtonian and non-Newtonian liquids*, Int. J. Multiphase Flow, vol. 65, 2014, 11-23.

[6] Böhm L., Brehmer M., Kraume M., *Comparison of the Single Bubble Ascent in a Newtonian and a Non-Newtonian Liquid: A Phenomenological PIV Study*, Chem. Ing. Tech., vol. 88(1-2), 2016, 93-106.

[7] Böhm L., Drews A., Kraume M., *Bubble induced shear stress in flat sheet membrane systems – Serial examination of single bubble experiments with the electrodiffusion method*, J. Membr. Sci., vol. 437, 2013, 131-140.

[8] Prieske H., Böhm L., Drews A., Kraume, M., *Optimised hydrodynamics for membrane bioreactors with immersed flat sheet membrane modules*, Desalin. Water Treat., vol. 8(1-3), 2010, 270-276.