Gas Phase Distribution in a Mixer with a Mechanical High-Speed Mixer – Case Study

Anna MŁYNARCZYKOWSKA[1)]

[1)] PhD, Department of Environmental Engineering, Faculty of Civil Engineering and Resource Management, AGH University of Krakow, Mickiewicza 30, Krakow, Poland; ORCID: 0000-0001-8072-5113, email: mindziu@agh.edu.pl

http://doi.org/10.29227/IM-2025-01-25

Submission date: 15-05-2025 | Review date: 29-06-2025

Abstract

The mechanical mixing process is a common operation in the technological processes of many industries, also in the mineral processing. The separation of valuable mineral from gangue in the flotation chamber results in electrical power consumption of 1-10 kW/m³ and depending on the type of flotation machine. Air is introduced into the flotation chamber and the bubbles are dispersed using a rotor-stator system, which simultaneously mixes the suspension with air bubbles.

Flotation efficiency depends on tank shape, rotor and stator design, as well as operating conditions such as rotor speed, aeration rate, and suspension properties. Striving to reduce energy consumption while maintaining high process efficiency, optimizing the shape of the stator-rotor system is crucial. This system determines the distribution of bubble in flotation chamber and the occurrence of elementary flotation acts.

The article presents experimental results from measurements of the velocity field of the water and air-water systems in a laboratory flotation chamber under various hydrodynamic conditions by using the digital image anemometry (PIV) technique and numerical simulations (CFD).The obtained dates were used to check the distribution of the gas phase in the flotation chamber and to assess the energy consumption of the rotor.

Keywords: *two-phase flow, mixer tank, high-speed rotor, PIV technique, CFD simulation*

1. INTRODUCTION

Examples of gas phase distribution in an aqueous medium can be found in many devices used in the process of sewage treatment, water regeneration or processing of fine-grained minerals like flotation. This is a process aimed at separating the useful component from the gangue of the feed using air bubbles in the aqueous medium as a carrier. The division criterion is the differences in the surface properties of the components selective enriched raw material (hydrophobicity or hydrophilicity of grains).

The important element of the mixer, which is the flotation machine, is the aerator (Fig. 1a) consisting of a rotor and a stator. The impeller is used to mix the flotation pulp, as well as to generate and disperse small air bubbles inside flotation chamber and transport aggregates to the foam layer on the surface this tank. The flotation air is supplied to the device through the rotor axis and flows out in the form of large bubbles inside the rotor zone.

The efficiency of froth flotation process is determined by many factors, because in the flotation cells complex multiphase flow phenomena occur, such as aeration of the flotation pulp, mixing of water, air bubbles and mineral particles; formation of permanent flotation aggregates; formation of flotation foam. One of the most important parameters is the design of the flotation machine, i.e. the shape of the rotor and flotation cell. It determines the hydrodynamic conditions of the multiphase suspension flow and energy consumption (the highest cost of the process).

Heuristic [1], phenomenological [2] and experimental models [3] used in the past to improve the design of flotation machines have been replaced by computer numerical calculations (CFD). The simulation results are verified using Particle Image Velocity (PIV) and technological parameters of the flotation process obtained from experiments [4, 5, 6]

The relationship between technological parameters and criterion quantities describing the hydrodynamics of the medium in the flotation process is possible by determining, among others, the velocity of gas bubbles or the rate of turbulent dissipation.

This manuscript presents the results of numerical calculations (CFD) and their experimental validation (PIV) in the scope of determining the velocity field structure for water and air-water systems in a laboratory flotation cell. The studies were carried out in various hydrodynamic conditions (rotor speed, air flow). The obtained data were used to check the gas phase distribution in the flotation cell and to assess the energy consumption of the mixing process.

2. EXPERIMENTAL SETUP

The experimental installation was a laboratory flotation cell with a WD-type aerator (vessel dimensions: (diameter – 0.18 m, height – 0.2 m). Water or a mixture of water and air bubbles was used as the experimental medium in the mixer. Measurements were carried out for the following rotor speeds: 350, 550, 750 and 1000 rpm, which correspond to Reynolds number values ranging from 74220 to 21257. The Particle Image Velocimetry (PIV) method was used to assess the velocity vector components. The marker particles were illuminated with a double Nd:YAG laser with an energy of about 40 mJ per pulse. Digital images were obtained using a 4 Mpx monochromatic CCD camera. The image recording time between two consecutive frames ranged from about 100 µs to 500 µs.

The Fig. 1 shows the important elements of the measurement station, i.e. a cylindrical chamber flotation placed in a

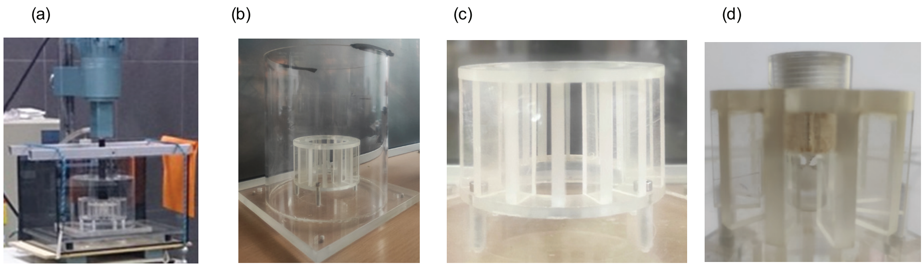

Fig. 1. Experimental elements: measurement system (a) flotation chamber (b), rotor calmer (c), WD impeller (d)
Rys. 1. Elementy eksperymentu: układ pomiarowy (a), komora flotacyjna (b), uspokajacz wirnika (c), wirnik WD (d)

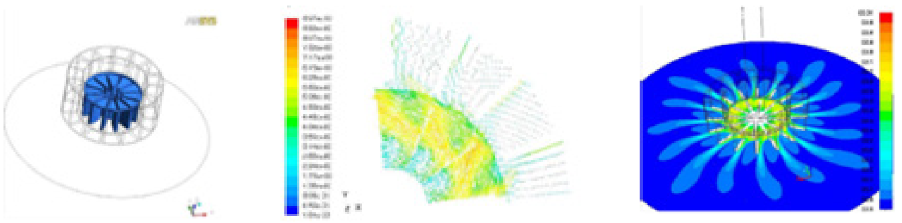

Fig. 2. Aerator WD project and velocities on a horizontal plane for (a) MRF model (b), and velocity distribution for 4E+6 elements coloured by velocity magnitude (c) [6]
Rys. 2. Projekt WD aeratora i prędkości na płaszczyźnie poziomej dla (a) modelu MRF (b) oraz rozkład prędkości dla 4E+6 elementów pokolorowanych według wielkości prędkości (c) [6]

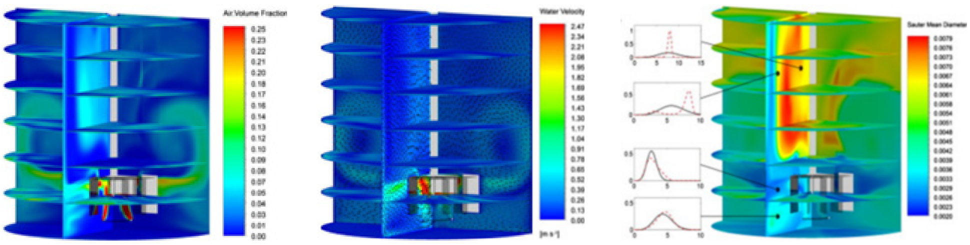

Fig. 3. Contour plots showing mean volume fraction of gas (a) and SMD (d32) and reconstructed normalized volumetric BSD (continuous line – normal distribution; discontinuous line) and mean liquid velocities and velocity vectors (c), for N = 350 RPM and Jg = 0.62 cm/s [8]
Rys. 3. Wykresy konturowe przedstawiające średni ułamek objętościowy gazu (a) i SMD (d32) oraz zrekonstruowany znormalizowany objętościowy BSD (linia ciągła – rozkład normalny; linia nieciągła) i średnie prędkości cieczy oraz wektory prędkości (c), dla N = 350 obr./min i Jg = 0,62 cm/s [8]

tank of water to avoid laser light reflections and the appearance of the tank and rotor.A detailed description of the research station is included in the earlier study [7]

2.1 Measurements Methodology

The flow pattern measurements for different impeller rotational speed and for various air flow rates by using the Particle Image Velocity (PIV) technique.

The measurements conducted in the axis perpendicular to the camera view (z=0cm) and in the plane z=3cm parallel from the axis plane. In the case of the measurement in the axis plane, due to light blinding, only the right half of the plane is well illuminated and the velocity vectors can was evaluated

3. RESULTS

3.1 CFD background for PIV measurements

The distribution of air bubbles in a pneumo-mechanical flotation chamber is critical to the efficiency and efficacy of the flotation process. Particularly concerning the parameters influencing air bubble size, distribution, and the overall hydrodynamic conditions in flotation chamber.

Szczygieł et al presented the results of modeling single-phase water flow in a flotation cell obtaining the structure of the water velocity field using the ANSYS Fluent CFD package. Computational tests have shown that without losing the quality of the results, in a short time, data can be obtained using the MRF method ("frozen rotor approach") for a mesh of polyhedral elements in the number of 700,000 (Fig. 1). The maximum calculated value of the velocity components was 0.74 m/s, and the experimental value was 0.75 m/s (chamber diameter 500 mm, rotor inner diameter 0.05 m, calmer diameter 0.85 m, rotor speed 500 rpm) [6.]

Basavarajappa and Miskovic [8], based on an extensive literature review, investigated the bubble size distribution in a flotation cell under specified hydrodynamic conditions and using the population equilibrium model (PBM) coupled with computational fluid dynamics (CFD). The researchers considered the phenomenon of bubble bursting and coalescence caused by turbulence. The interphase interactions were limited to the model of the resistance to motion caused by the presence of gas. Other forces acting at the phase boundary were omitted. The obtained results of direct measurements and numerical modeling were correlated. The Fig. 3 presents selected analysis results in relation to the volume fraction of the gas phase, the distribution of air bubbles expressed by the mean Sauter diameter (SMD d32). The highest gas and water velocity was 2.47 m/s in the closed rotor area, while in the calm flow zone (area above the rotor up to 1/2 tank height) it

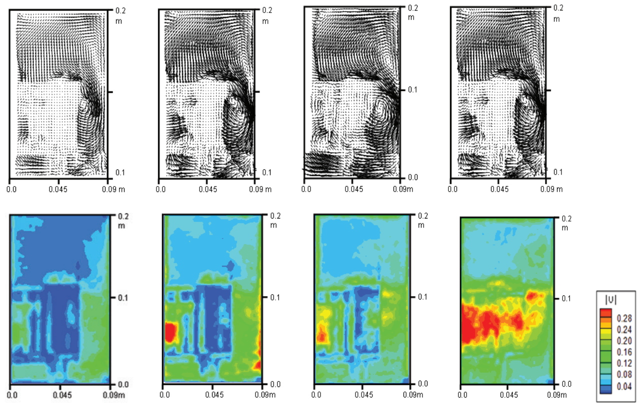

Fig. 4. Velocity fluctuation field on vertical plane for (a)350, (b)550,(c)750 and (d)1000 rpm without gas flow. Contours of stream-wise velocity in the blind drift in the x-y cross section

Rys. 4. Pole fluktuacji prędkości na płaszczyźnie pionowej dla (a)350, (b)550, (c)750 i (d)1000 obr./min bez przepływu gazu. Kontury prędkości wzdłuż strumienia w dryfie ślepym w przekroju x-y

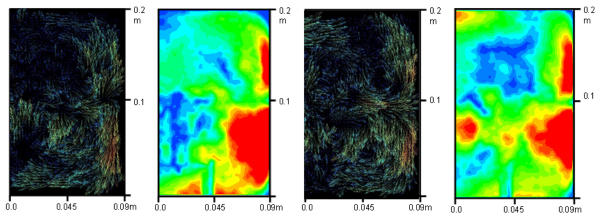

Fig. 5. Velocity vectors and contours in plane z=3cm for (a), (b) 1000rpm, gas flow 24l/min, (c), (d) 1000rpm, gas flow 240 l/min. the channel ending

Rys. 5. Wektory prędkości i kontury na płaszczyźnie z=3cm dla (a), (b) 1000 obr./min, przepływu gazu 24 l/min, (c), (d) 1000 obr./min, przepływu gazu 240 l/min. zakończenie kanału

ranges from 1.04 to 1.56 m/s. This corresponds to the recirculation regime.

Additionally, using the indicated CFD-PBM methodology, it is also possible to estimate the distribution of air bubble sizes at different heights of the flotation cell.

3.2 CFD and PIV results and discussion

Fluid motion is the first essential component in the attempt to describe hydrodynamics in a flotation cell. It is highly turbulent and unstable. A complete picture of the flow velocity structure and all fluctuations of its value obtained from PIV measurements and CFD simulations, allows for characterizing the operation of the mixer.

The velocity fluctuation field in the vertical center plane for 350, 550, 750 and 1000 rpm without gas flow is shown in Figures 4 (second line).

Comparing the velocity vector images, one can see clear differences between the fluctuations in different regions of the rotor zone. The area with low velocity fluctuation is located above the calmer-rotor system. The area with medium velocity fluctuation is located below the calmer, and the area with the highest velocity fluctuations is located between the plane of the calmer-rotor system and the wall of the flotation cell.

The images of the velocity vectors of the field in the vertical plane (Fig. 4 – first line) show that the liquid above the rotor-stator system is pumped to the upper surface near the tank wall, while in the area away from the wall the direction is opposite.

The movement of the rotor causes significant liquid circulation. The flow pattern in the space between the stator and the chamber wall is opposite to the pattern above the upper surface of the rotor.

Most of the area below the rotor-stator system, the liquid moves along the radius. The exceptions are a small area near the wall where the liquid is directed to the bottom of the tank and the center of the flotation cell, where the liquid moves from the bottom to the top. Due to the design of the stator-rotor system, accurate measurements of the velocity vectors are limited. This can cause discrepancies in comparison with the CFD results.

It should be clearly indicated that the PIV measurements confirm the radial and axial flow of the liquid in the positive direction with the rotor speed. The above observations are confirmed by the studies of Szczygieł et al. [6]

The velocity vector fields for the cases with air flow of 24 and 240 l/min and 1000 rpm of the rotor speed are shown in Figures 5a and Figures 5b, respectively. The aeration of the fluid and generation of air bubbles resulted in the amplification of the flow in the rotor-calmer zone.

The increase in air flow (240 l/min) causes the formation of two local turbulences (vortices) and the change of the medium flow direction. Below the rotor, the flow has a low velocity and is stable, while outside the rotor-calmer system and close to the tank walls, the flow velocity is strong and up to 10 times thicker compared to the case without gas flow.

This is a general trend for all the cases studied (Fig. 6). The highest value of the flow velocity was 0.53 (m/s) and was recorded for the maximum rotor revolutions and the highest air dose. The lowest was 0.02 (m/s).

4. CONCLUSION

This paper presents the structure of the mean velocity fields in the cross-sectional plane of the flotation cell and in

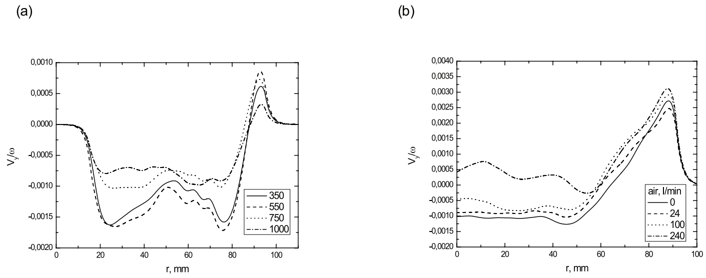

Fig. 6. Flow stream-wise for the ca Fluid velocity vertical component at the cross section 1cm above the impeller:(a)for various impeller speed and without the air flow,(b)for different air flow rates and for impeller speed 1000rpm. ses (1), (3) and (4) at a distance y = 30 cm from the fan outlet [7]

Rys. 6. Przepływ w kierunku strumienia dla ca Pionowa składowa prędkości cieczy na przekroju poprzecznym 1 cm nad wirnikiem: (a) dla różnych prędkości wirnika i bez przepływu powietrza, (b) dla różnych szybkości przepływu powietrza i prędkości wirnika 1000 obr./min. sekcje (1), (3) i (4) w odległości y = 30 cm od wylotu wentylatora [7]

the region of 1 cm above the rotor for specific hydrodynamic conditions, i.e. rotor speed and cell aeration. These results were obtained using the digital imaging particle velocity method.

CFD simulations and PIV measurement data confirm that with the increase of gas flow and rotor speed, the flow velocities of the dispersed phase (liquid) and dispersed phase (gas) also increase. This confirms the turbulent nature of the flow, especially in the space between the rotor calmer and the cell wall.

It should be noted that in the aerator region there is no flow turbulence for the speed of 750 rpm. Vortices between the stator barriers occur at the speed of 1000 rpm and above.

CFD modeling allows determining the air hold-up and air bubble distribution, the size of air bubbles at different heights from the bottom of the tank. Direct measurements using PIV, in combination with simulations of phenomena occurring in the area around the flotation machine aerator, enable the description of flows in the flotation machine working chamber.

In summary, the use of CFD modeling supported by PIV measurements is helpful in terms of optimizing the aerator design, in accordance with the established operating parameters.

ACKNOWLEDGEMENTS

The present work was supported by Research project partly co-financed by the program “Excellence Initiative-Research University” for the AGH University of Krakow (ID 5186)

Literatura – References

1. Brożek, M., Młynarczykowska, A. (2012). The distribution of air bubble size in the pneumo-mechanical flotation machine. Archiv Min. Sci., 57(3), 729–740. doi 10.2478/v10267-012-0047-9
2. Młynarczykowska, A., Oleksik, K., Tupek, K Dispersion of air bubbles in a flotation cell under predetermined hydrodynamic conditions. E3S Web of Conferences. 8, 01012, 1. ISSN 2267-1242. doi: 10.1051/ 3 e sconf/2016 0 01012
3. Brożek, M., Młynarczykowska, A. (2007).Analysis of kinetics models of batch flotation. Physicochemical Problems of Mineral Processing. 41, 51-35. ISSN: 0137-1282
4. Koh, P.T.L, M.P. Schwarz, M,P.(2007). CFD model of self-aerating flotation cell. Int. J. Mineral Process. 85: 16–24. https://doi.org/10.1016/j.minpro.2007.08.006
5. Tiitinen, J., Vaarno, J., Grnstrand, S. (2003). Numerical modeling of an Outokumpu flotation device. 3rd Int. Conf. on CFD in the Minerals and Process Industries, Melbourne, Australia
6. Szczygiel, I., A. Fic, A., Sachajdak, A., M. Rojczyk, M., Buliński, Z.,Mańka, A. (2013). Numerical simulation of single phase flow in a flotation machine. Computer Assisted Mechanics and Engineering Sciences. 20: 145–156. . doi: 10.5151/meceng-wccm2012-19492
7. Jaszczur M., Młynarczykowska A., Hanus R. (2016). An analysis of the velocity field distribution inside the flotation chamber. Journal of Physics: Conference Series 745, 032121. doi:10.1088/1742-6596/745/3/032121
8. Basavarajappa, M. , Miskovic, S. (2016). Investigation of gas dispersion characteristics in stirred tank and flotation cell using a corrected CFD-PBM quadrature-based moment method approach. Minerals Engineering 95, 161–184. http://dx.doi.org/10.1016/j.mineng.2016.06.026

Dystrybucja fazy gazowej w mieszalniku z mechanicznym mieszadłem szybkoobrotowym – studium przypadku

Proces mechanicznego mieszania jest powszechną operacją w procesach technologicznych wielu gałęzi przemysłu, również w przetwórstwie minerałów. Oddzielenie cennego minerału od skały płonnej w komorze flotacyjnej powoduje zużycie energii elektrycznej wynoszące 1-10 kW/m³ i w zależności od rodzaju maszyny flotacyjnej. Powietrze jest wprowadzane do komory flotacyjnej, a pęcherzyki są rozpraszane za pomocą układu wirnik-stojan, który jednocześnie miesza zawiesinę z pęcherzykami powietrza. Sprawność flotacji zależy od kształtu zbiornika, konstrukcji wirnika i stojana, a także warunków pracy, takich jak prędkość wirnika, szybkość napowietrzania i właściwości zawiesiny. Dążąc do zmniejszenia zużycia energii przy jednoczesnym zachowaniu wysokiej sprawności procesu, optymalizacja kształtu układu stojan-rotor ma kluczowe znaczenie. Układ ten określa rozkład pęcherzyków w komorze flotacyjnej i występowanie elementarnych aktów flotacji.

W artykule przedstawiono wyniki eksperymentów z pomiarów pola prędkości układów woda i woda-powietrze w laboratoryjnej komorze flotacyjnej w różnych warunkach hydrodynamicznych, wykorzystując technikę cyfrowej anemometrii obrazowej (PIV) oraz symulacje numeryczne (CFD). Uzyskane dane posłużyły do sprawdzenia rozkładu fazy gazowej w komorze flotacyjnej oraz do oceny zużycia energii przez wirnik.

W artykule przedstawiono wyniki eksperymentów z pomiarów pola prędkości układów woda i woda-powietrze w laboratoryjnej komorze flotacyjnej w różnych warunkach hydrodynamicznych, wykorzystując technikę cyfrowej anemometrii obrazowej (PIV) oraz symulacje numeryczne (CFD). Uzyskane dane posłużyły do sprawdzenia rozkładu fazy gazowej w komorze flotacyjnej oraz do oceny zużycia energii przez wirnik.

Słowa kluczowe: *przepływ dwufazowy, zbiornik mieszający, wirnik szybkoobrotowy, technika PIV, symulacja CFD*