An analysis of the velocity field distribution inside the flotation chamber

Marek Jaszczur, Anna Młynarczykowska, Robert Hanus

1 Department of Fundamental Research in Energy Engineering, AGH University of Science and Technology, Poland
2 Faculty of Mining and Geoengineering, AGH University of Science and Technology, Krakow, Poland
3 Rzeszów University of Technology, Rzeszów, Poland

Email: mindziu@agh.edu.pl

Abstract. This paper presents the experimental data of the measured velocity field of a water and air-water system within a laboratory flotation chamber at various rotational speeds of the rotor and aeration. These results were used to verify the gas-phase distribution in the flotation chamber of a process conducted under specified hydrodynamic and physicochemical conditions. The PIV technique was used to determine the flow field in a number of planes; with this method non-invasive measurements of the velocity field in single and two-phase systems were carried out. An analysis of the results allows one to determine the parameters describing the hydrodynamic conditions in the flotation chamber and evaluate the effectiveness of the flotation process.

1. Introduction

Flotation is an important process broadly applied in the mineral industry. The capacity of a typical plant exceeds a thousand tons per hour and the flotation chamber range in volume up to 300m³. The energy required for this industrial process is typically very large. In order to separate the valuable minerals from the gangue materials in a flotation chamber, an electrical power input of the order of 1-10kW/m³ is required. The valuable mineral particles are separated from the ore when they are attached to the air bubbles and they float to the top surface via a froth layer. The injection and dispersion of the air bubbles are performed with a rotor-stator device, which also mixes the slurry with the air bubbles.

The process performance is evaluated in terms of the recovery of the minerals in concentrate and the minerals left in the tailings. The process performance is influenced by several geometrical features, such as the tank geometry, the impeller and stator design, as well as by the operating conditions, such as the impeller rotation speed, the aeration rates and the slurry properties. Yet one of the key elements in the flotation cell is the rotor-stator, which mixes the slurry with the air bubbles. Therefore, a proper optimization of this component may significantly improve the flotation process performance and decrease the process energy demand. In order to model this process successfully, it is very important to understand the phenomena occurring in the flotation process as well as evaluated model input process parameters on it. Arbiter and Harris [1] found that the froth zone and the pulp zone should be considered separately. Runge et al.[15] and Schwartz et al.[17] proposed a floatability component approach in which the feed is divided into groups of particles having the same response and during the analysis, the mass fraction coefficient of a specific mineral is calculated as a mass weighted recovery of each component group. Another one was proposed by Savassi [16], in which the recovery by entrainment is calculated based on the recovery of the water to the concentrate and the ratio of the
recovery of the liberated hydrophilic solids to the recovery of the water to the concentrate. The
extension of the basic models for the calculation of the pulp phase rate constant takes into account the
energy dissipation and the kinetic energy of turbulence [13,22].

In the last two decades, researchers have studied the flotation process using very powerful tools -
computational fluid dynamics (CFD) - which provide details of the flow phenomena. The flotation
process, in order to be fully modelled, requires a good selection of the turbulence model as well as the
multiphase model. Three dimensional rotating grid modelling with such models requires a very large
rotating grid and, in consequence, large computing power. A comparative analysis of the velocity field
in a flotation cell was performed by Shi et al.[18], while the influence of the turbulent models on the
flow field in a flotation cell was analyzed by Xia et al. [23]. In the last decade, a large number of
challenges encountered in the modelling of stirred vessels have been reported in literature [5,9-11].

The relation between the experimental measurements in the laboratory flotation cell and the
hydrodynamic conditions in the flotation machine is possible to be determined by way of describing
the hydrodynamics with the use of a series of parameters, i.e. the bubble velocity and the turbulent
dissipation rate. For example, the energy dissipation rate is obtained as the ratio of the power input to
the mass of the fluid within the chamber. An example can be the research of Karimi et al.[6,7], in
which the possibility of modelling -predicting the constant value of the galena and chalcopyrite
flotation kinetics was verified based on the methodology elaborated by Pyke et al.[12]. The capability
of this approach is demonstrated as a CFD-based methodology for flotation modelling.

Due to the shape of the flotation chamber, the rotor construction and the working conditions of the
flotation machine, a specific type of hydrodynamic conditions can be present in the working chamber,
which significantly influence the process of air bubble mineralization (formation of flotation
aggregates) and the effectiveness of the whole froth flotation process. We can distinguish between: the
turbulent flow in the aerator space, the gas phase dispersions in the two-phase liquid-gas system, the
solid phase dispersion in the liquid-solid system, the flotation of the gas bubbles and mineral grains to
the top of the chamber and the formation of flotation forth for the three-phase liquid-gas-solid system
(see Fig.1). The necessary condition for the occurrence of the flotation process is a strong connection
between the three mentioned phases.

The flow phenomena taking place inside the flotation chamber with the participation of phases of
different densities, caused by the work of the movable parts of the flotation machine, makes the
investigations and modelling of this process very difficult. For the mentioned reasons, analogous
problems are faced by constructors of flotation machines, whose work effectiveness has been, and is,
verified based on the results of flotation tests realized in the laboratory and technical scale.

Szczygiel et al. [19] proposed the use of PIV to determine the distribution of the velocity fields
inside the flotation chamber for the validation of the mathematical model of the flotation process,
based on the dispersion of the disperse phase in the two-phase system (CFD). The mentioned test
methodology constitutes an alternative for the time-consuming experimental verification of the
flotation machine construction solutions. The PIV technique for the two-phase flow was also used by
Tiger and Pan [20], who applied the filtering technique in order to separate the phases. PIV with a
fluorescent dye was used by Bröder and Sommerfeld [2], who analyzed the fluid flow on a bubble
column. This measurement technique was utilized to study the flotation cell by many researchers.

Based on the mentioned examples, as well as other literature reports describing the use of CFD
modelling techniques [8,14] and PIV flow analysis techniques [4,21], we can conclude their high
usefulness for the description of the hydrodynamic conditions present in the working chamber of the
flotation machine. In this paper, the fluid flow phenomena in the flotation cell were studied with the
use of particle image velocimetry measurements.

2. Experimental setup and procedures

The experimental set-up shown in Figure 2 represents a laboratory set-up of the stirred vessel
equipped with a pitch blade turbine. The geometrical scale of the physical model was as follows: the
diameter of the vessel 0.18 m, the height of the chamber 0.2 m. It was assumed that the water and air
flows are steady and isothermal. The water, or the mixture of water and air bubbles, was used as the experimental fluid in the stirred vessel. The diameter of the impeller was half of the vessel diameter and it was placed at a distance of about one-fifth of the vessel diameter from the bottom. In order to minimize the optical distortion, a cylindrical vessel was installed in a rectangular vessel filled with distilled water. The cylindrical vessel was filled with water to the height $z=0.1m$. The speed of the impeller was set-up to 350, 550, 750 and 1000 rpm, which corresponds to the Reynolds number varying from 74220 to 21257.

The Particle Image Velocimetry (PIV) method was used to evaluate the velocity vector components. The particles were illuminated with a double-pulse Nd:YAG laser of the energy of about 40 mJ per pulse. The digital images were acquired by a 4 Mpx monochromatic CCD camera. In each experiment, 100 double frame images were recorded with the camera recording at a frequency of 5 Hz, which resulted in the overall time of one measurement of around 20 seconds. An analysis of the influence of the number of images on average value has been tested and the value of 100 frames was sufficient in order to obtain the first and second order statistics.

The time $\Delta t$ between two subsequent frames varied from about 100 $\mu s$ to 500 $\mu s$. However, the measurements performed locally inside the cavity were made in the range of 3000-4000 $\mu s$, because the local velocities were much lower than those in the other regions. In order to evaluate the velocity vectors, the cross-correlation method was used and the interrogation window from 32x32 to 16x16 with the overlap of 25% was applied. With the purpose to stabilize the temperature before the experimental measurements were performed, the impeller rotated in the vessels for about 60 minutes. For the measurement with pure water without air bubbles, glass hollow spheres seeding was used with the diameter of about 1 $\mu m$. In the cases of a two-phase air-liquid flow, small air bubbles acted as the seeding particles.

The examinations were realized with the use of a test stand consisting of: a laboratory pneumo-mechanical flotation machine with a cylindrical WD-type chamber (see Figure 2); a flotation machine control system (adjustment of impeller speed, amount of injected air etc.).

The measurements were 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 be evaluated.

![Flotation Process](image1.png)

**Figure 1.** Flotation Process (left) and aerator model of the flotation cell (right).
3. Experimental set-up and procedures

The flow motion generated by the impeller and bubbles impact the flotation process characteristics. For this reason the flow pattern for different impeller rotational speed and for different air flow rates were investigated using the PIV technique.

The flow pattern in the central section of the flotation chamber is presented in the Figure 3. The velocity vectors field on vertical plane are shown in the cases without air flow (no bubbles were generated by additional air) and for the impeller rotational speed 350, 550, 750 and 1000rpm. It can be seen that the fluid above the impeller-stator in the area close to the chamber wall is pumped up to the top surface while in the region far from the wall direction is opposite. The flow movement generates large fluid recirculation. The flow pattern in the space between the stator and chamber wall is the opposite to the pattern above impeller top surface. The velocity vector axial component is also very strong in the area close to the wall but now the fluid direction is up to the bottom of the flotation cell. In the most of the area below the impeller-stator devices fluid has mainly radial component except the small area close to the wall and the geometrical center of the flotation cell where the fluid is moving to the or out of the bottom respectively. It has to be noticed that some places below the stator (around \( r=0.045 \text{m} \)) are not fully visible due to filament distortion also the area very close to the axis center is occupied by a shaft and this area is also not visible. In general flow motion in this plane with the upper and lower fluid recirculation may coarse suspension as well aid mineral particles mixing. The PIV measurements show that the radial and axial velocity positively correlates with impeller velocity. Increasing the impeller rotational speed enhance fluid motion, increasing the fluid velocity and shifting slightly position of the fluid recirculation center.

In the flotation process where the fluid motion is unsteady, not only the average fluid velocity pattern is important but also fluctuation of the velocity vectors. This allows evaluate the area where the transport phenomena will be enhanced. Velocity fluctuation field in the vertical center plane for 350, 550, 750 and 1000rpm without gas flow is shown in Figure 4. In the velocity fluctuation figure the differences between fluctuations in different regions can be easy seen. The area with small velocity fluctuation located above the stator-impeller. The area with medium velocity fluctuation located bellow the stator and the area with the highest velocity fluctuations located between the stator-impeller wall and the flotation chamber wall. The location where the stator and the impeller are installed has been measured and is presented in the figure, but stator-impeller location, it is not fully
representative because due to optical reflection and distortion it is hard to distinguish between impeller velocity and fluid flow velocity.

![Figure 3](image-url)

**Figure 3.** Velocity field on vertical plane for (a)350, (b)550,(c)750 and (d)1000 rpm without gas flow

The flow pattern in the plane $z=3cm$ of the flotation cell for the cases with the air flow is presented in the Figure 5. The velocity vectors field 5(a) and 5(b) are shown for the cases with air flow 24 and 240 l/min respectively and for the impeller rotational speed 1000rpm. It can be seen that the fluid motion in the area above the stator is significantly enhanced by the bubbles’ motion. For the air flow rate 240l/min fluid motion in the area near the center axis changes direction. Instead of one large vortex generated in this area flow is more complicated and two large vortices are observed. In this case large area above the stator has very low velocity. At the same time the near wall region with a large axial component of the velocity is much thicker (in some cases 10 times thicker in reference to the cases without air flow rate). This is seen, for all air flow rates considered here. Increasing amount for air delivery to the flotation chamber significantly influenced the axial component of the velocity in the near center region. At a selected air flow rate (not presented here) it is even possible to stop the fluid
motion (and in consequences hydrotransport) in this area of the flotation chamber. Adding air bubbles significantly increase fluid motion in the area between the stator and chamber wall as well as in the area below the stator. It is worth to notice, that fluid motion enhancement in analysed range is almost air flow rate independent. The area with low and high velocity magnitude is well seen as a contour plot of velocity magnitude presented in figure 5(c) and 5(d). In Figure 6 the fluid velocity vertical component normalized by the angular velocity \( \omega \) of the impeller at the cross section 1cm above the impeller is presented. The influence of the various impeller speed on velocity axial component for the cases without the air flow is shown in figure 5(a) and for constant impeller speed 1000rpm but different air flow rates 0, 24, 100, 240l/min is shown in figure 5(b).

**Figure 4.** Velocity fluctuation field on vertical plane for (a)350, (b)550,(c)750 and (d)1000 rpm without gas flow.
Figure 5. Velocity vectors and contours in plane $z=3\text{cm}$ for (a),(c) $1000\text{rpm}$, gas flow $24\text{l/min}$, (b),(d) $1000\text{rpm}$, gas flow $240\text{ l/min}$.

Figure 6. Fluid velocity vertical component at the cross section $1\text{cm}$ above the impeller: (a) for various impeller speed and without the air flow, (b) for different air flow rates and for impeller speed $1000\text{rpm}$.
4. Conclusion
In the present paper, digital particle image velocimetry method was used to determine the mean velocity fields in the flotation cell cross-section plane and in the area 1cm above the impeller. The measurements were performed under specified hydrodynamic conditions, i.e. impeller speed and velocity fields in the flotation cell cross-section plane and in the area 1cm above the impeller. The

In the present paper, digital particle image velocimetry method was used to determine the mean velocity fields in the flotation cell cross-section plane and in the area 1cm above the impeller. The measurements were performed under specified hydrodynamic conditions, i.e. impeller speed and velocity fields in the flotation cell cross-section plane and in the area 1cm above the impeller. The distribution of the mean air velocities confirms that, together with the increase of the gas flow and impeller speed, the flow velocities of the disperse phase and the dispersed phase (gas) increase as well. A significant increase of the flow velocity of both phases, proving the turbulent character of the flow, is observed, especially in the space between the impeller stator and the cell wall. This is caused by the directed flow of the medium from among the stator in the form of fluxes ejected into the space beyond the aerator.

The movement of the two-phase medium is described by the velocity vectors, whose direction is in accordance with the movement observed visually. The results show that, together with the increase of the impeller speed, the fluid velocity value increases as well, and the distribution of the velocity vectors illustrates their movement. It should be noted that in the aerator area, there is no flow turbulence for the speed of 750 rpm. The vortexes between the stator barriers are present for the speed of 1000 rpm and above. The direct measurements with the use of PIV, combined with the CFD occurring in the area around the aerator of the flotation machine, make it possible to describe the flows in the working chamber of the flotation machine. They also allow us to draw conclusions in reference to the optimization of the aerator construction, according to the established working parameters.

Acknowledgments
The present work was supported by the Polish Ministry of Science (Grant AGH No. 11.11.210.312).

References
[1] Arbiter N, Harris CC, 1962 Froth Flotation. Flotation kinetics. In: Fuerstenau, D.W. (Ed.), American Institute of Mining, Metallurgy and Petroleum Engineers, New York, pp. 215–246.
[2] Brüder D and Sommerfeld M, 2001 4th Int. Symp PIV, Göttingen, Germany, 17–19 September.
[3] Cornelius F, Vos CF, Stange W, Dei Indre J, Bradshaw D 2014 Minerals Eng. 66, pp. 62-67.
[4] Hoque M, Sathe M J, Mitra S, Joshi J B, Evans GM, 2015. Chem. Eng. Sci. 137 752–767.
[5] Joshi J B, Nere NK, Rane C V, Murthy B N, Mathpati C S, Patwardhan A W, Ranade V V, 2011 Can. J. Chem. Eng. 89, 23–82.
[6] Karimi M, Akdogan G, Bradshaw S M, 2014 Minerals Engineering. 69, pp. 205–213.
[7] Karimi M, Akdogan G, Bradshaw S M, 2014 Minerals Engineering. 69 ,pp. 214–222.
[8] Koh P T L, Schwarz M P, 2007 Int. J. Mineral Process. 85, pp. 16-24.
[9] Lane G, Schwarz M, Evans G, 2005 Chem. Eng. Sci. 60 (8), pp. 2203–2214.
[10] Malyśa K, Cymbalisty S, Czarnecki L J, Masliyah J, 1999 J. Miner. Process. 55, pp.171–188.
[11] Mańka A, Buliński Z, Szczygiel I, Fic A, 2012 X Miedzynarodowa Konferencja Przeróbki Rud metali Nieżelaznych INOC”2012 Trzebiezowice, 17-19 Października 2012(in polish).
[12] Murthy B, Ghadge R, Joshi J, 2007Chem. Eng. Sci. 62, pp.7184–7195.
[13] Pyke B, Fornasier D, Talston J, 2003 J. Colloid Interface Sci. 265, pp.141–151.
[14] Ranade V V, Tayalia Y, Krishnan H, 2002 Chem. Eng. Commun. 189, pp.895–922.
[15] Runge K C, McMaster J, Ijsselstijn J, Vien A, 2007 Flotation ’07, Cape Town, South Africa.
[16] Savassi O N, 1998 Ph.D Thesis. J. Kruttschnitt Mineral Research Centre, Univ of Queensland.
[17] Schwarz S, Alexander D, Whiten W J, Franzidis J P, Harris MC, 2006 In: XXIII International Mineral Processing Congress Proceedings, Istanbul, Turkey.
[18] Shi S, Zhang M, Fan X, Chen D, 2015 International Journal of Mineral Processing 142,pp.2–9.
[19] Szczygiel I, Fic A, Sahajdak A, Rojeczyk M, Mańka A, 2013 CAMES, 20, pp.145–156.
[20] Tiger K T, Pan C, 2000 Journal of Fluids Eng. 122, pp.811–818.
[21] Virdung T, Rasmuson A, 2007 Chemical Engineering Research and Design 85, pp.193–200.
[22] Yoon R H, Kelley K, Do H, Sherrell I, Nobel A, Kelles S, Soni G, 2012 XXVI International Minerals Processing Congress, New Delhi, India, pp.5969–5982.
[23] Xia J, Rinne A, Grönstrand S, 2009. Miner. Eng. 22, pp.880–885.