Blucher Mechanical Engineering Proceedings
May 2014, vol. 1 , num. 1
www.proceedings.blucher.com.br/evento/10wccm

NUMERICAL ANALYSIS OF THE WATER AERATION IN THE FLOTATION MACHINE

I.Szczygiel , A.Fic, A.Sachajdak, M.Rojczyk' Z.Buliński

Institute of Thermal Technology, Silesian University of Technology, Gliwice, Poland, (iszczygiel@polsl.pl)

Abstract. *Numerical model of the flow phenomena in the flotation machine is presented in the paper. The process of flotation consists of a number of phenomena which provide serious numerical difficulties. These include rotation, multiphase flow, foam formation etc. To the knowledge of authors, there is no complete numerical model available for the flotation machine. The long term task of the project is to develop the model of processes taken place in the flotation machine. Such a model would be very helpful designing these machine or during their modernization. In the paper, a two-phase flow of water and air in the flotation cell is analysed. For this purpose the commercial package ANSYS-Fluent was employed. The Interfacial Area Concentration method was used to model the two phase flow.*

Keywords: *Flotation, multiphase flow, numerical analysis, interfacial area concentration*

1. INTRODUCTION

Flotation is a process commonly used in various branches of industry to separate one constituent of minerals from the other. The flotation process of the water suspension of fine minced minerals is accomplished by the flotation machines (flotation cell, figure 1), which operate based on the differences of wettability of mineral particles. The main element of the machine is aerator (figure 2) consisting a rotor and stator, which are located in a special tank. The rotor is applied for mixing the water suspension of fine minced minerals (flotation pulp), as well as for dispersion of the flotation air in the pulp. The flotation air is delivered to the machine by the rotor axis. The air bubbles flowing in the pulp collide with mineral particles and create the aggregates with particles of selected minerals (hydrophobic particles attached to bubbles) called the flotation froth. The froth is lighter than water. As a result, material of lower wettability is transferred by the froth to the surface of the flotation pulp and removed by skimming device.

Complex multiphase flow phenomena take place in the considered flotation cells, namely aeration of the flotation pulp, mixing of water, air bubbles and mineral particles, creation of the flotation froth and its separation from the bulk mixture. The solid, liquid and gaseous phases interact with each other and they all participate in the flotation process. Modelling such a process was impossible in the past. Designing the flotation machines was based on empirical formulas and experience of the designers.

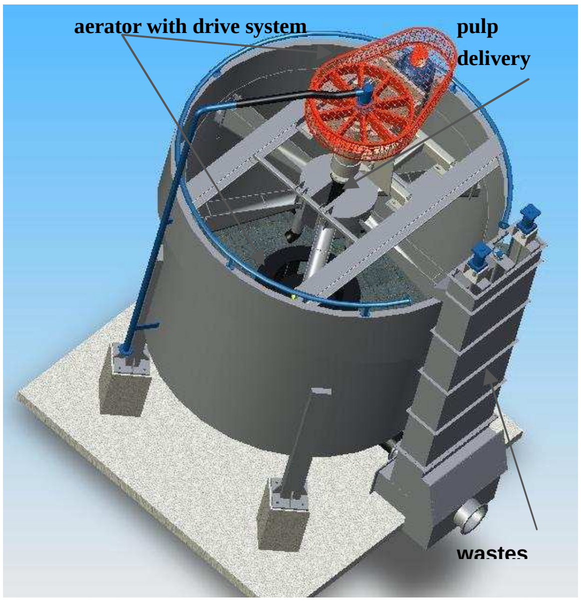

Figure 1. Typical flotation machine.

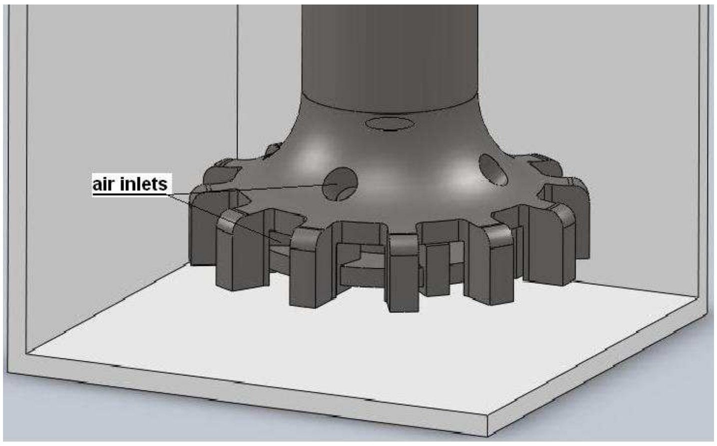

Figure 2. Aerator of the Denver flotation machine modelled in the paper.

Nowadays, numerical simulations of flotation phenomena have become possible due to sufficiently powerful computational machines, which allow to employ for this purpose sophisticated Computational Fluid Dynamic (CFD) packages, e.g. FLUENT or CFX. First attempts to modelling the flotation process using the CFD packages can be noticed in the last decade. The results of those are reported in literature, e.g. [2, 3, 5, 7]. There are even some thematic sessions regarding the simulations of the flotation phenomena (e.g. [7]) organized during the conferences devoted to numerical modelling multiphase industrial flow processes. Unfortunately, these first attempts provide rather simplified models of partial phenomena taking place during the flotation. Thus, there are still a lot of problems to be solved in this field. Due to mentioned complexity of phenomena, modelling the flotation process using the CFD is still a very challenging task.

The paper deals with preliminary results of modelling the processes that take place in a flotation cell. At this stage they relate to the aeration process of water in the flotation cell, i.e. two-phase flow of bubbles in mixed water is considered. Presented figures show selected fields of velocity and air volume fraction obtained in the domain of the aerator of the experimental flotation machine model. *ANSYS-Fluent* [1] CFD package was employed in the investigations. The computations were performed using the Multiple Reference Frame (MRF) model of rotating elements, as well as the Interfacial Area Concentration (IAC) [4] model for the transport phenomena of the air-water mixture. The numerical model was validated using PIV measurements.

2. MATHEMATICAL MODEL

The fluid flow in the flotation machine experiences complex physics, and this reflects in the mathematical expressions. The features of the flow are:

- three dimensional flow,
- turbulent flow,
- multiphase flow,
- unsteady state

The phenomena in the flotation machine are described by:

- continuity equation,
- momentum equation,
- turbulence model,
- rotation model,
- species transport model,
- foam formation model.

At this stage of investigations only the aeration process of water in the flotation cell is considered. The multiphase flow is simplified to two phases, namely water and air.

Taking into consideration the above simplifications the governing equations can be presented as follows:

- continuity equation;
- momentum equation:
- multiphase flow description;
- turbulence model.

It is assumed hereafter that all distinguished phases are mathematically treated as interpenetrating continua. Since the volume of a phase cannot be occupied by the other phases, the fields for all variables and properties are shared by the phases and represent volume-averaged values, as long as the volume fraction of each of the phases is known at each location. Volume fractions (α_q) represent the space occupied by each phase, and the laws of conservation of mass and momentum are satisfied by each phase individually.

The volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all the phases. These equations are closed by providing constitutive relations that are obtained from empirical information. Based on the local value of α_q, the appropriate properties and variables will be assigned to each control volume within the domain.

In the momentum conservation equation the phenomena specific to multiphase flows, such as lift forces, virtual mass forces or interaction forces between phases should be considered.

The size and distribution of the discrete phase can change rapidly due to mass transfer between phases, expansion or compression due to pressure changes, coalescence, breakage and nucleation mechanisms. The Population Balance model [6] ideally captures these phenomena, but due to the need of solving several transport equations it is computationally expensive. The interfacial area concentration model seems to be attractive the alternative to population balance model. It uses single transport equation per secondary phase. It can be especially recommended for bubbly flows in liquid, so phenomena in floatation machine it suits perfectly.

The transport equation for the interfacial area concentration can be described as follows [1, 4]:

$$\frac{\partial \rho_g \chi_p}{\partial \tau} + \nabla\left(\rho_g \vec{u}_g \chi_p\right) = \frac{1}{3}\frac{D\rho_g}{D\tau}\chi_p + \frac{2}{3}\frac{m_g}{\alpha_g}\chi_p + \rho_g\left(S_{RC} + S_{WE} + S_{TI}\right) \tag{1}$$

were χ_p is the interfacial area concentration, α_g is the gas volume fraction, ρ_g stands for the gas density, $\vec{u}_g$ for its velocity, $\dot{m}_g$ is the mass transfer rate into the gas phase per unit volume. S_{RC} and S_{WE} are the coalescence sink terms due to random collisions and wake entrainment, respectively. S_{TI} is the breakage source term due to the turbulence impact. The source terms can be evaluated using Hibiki-Ishii or Ishii-Kim models.

ANSYS-Fluent 13.0 commercial package was employed to solve the system of equations. The solver utilizes control volume method to convert the governing differential equations into algebraic ones.

3. MODEL VALIDATION

Multiphase models of complex processes require always validation. This can be accomplished comparing results of simulations against experimental ones. For this purpose, a physical model of flotation machine was build. The velocity measurements were performed with the utilization of PIV (Particle Image Velocimetry) technique. To make it possible, the container of laboratory machine was made from transparent material. The construction of the model allows for flexible adjustment of both velocity of the rotor and the amount of delivered air bubbles. The measurement plane is lightened by the laser lamp which allows for the observations of fluid with seeds or bubbles with the high speed camera. The pictures taken from the camera are then processed with a sophisticated software which outputs the distributions of fluid velocity in the selected area. The laboratory stand is shown in Figure 3 and the selected measurement plane is presented in Figure 4. The presented next velocity fields relate to this plane.

Using the technique mentioned above, the measurements of fluid velocity were qualitatively and quantitatively compared with the distributions obtained from the numerical model. The velocity measurements were averaged in the time interval of 3s. The results in the form of the projection of velocity vectors onto a horizontal frame are presented in Figure 5. The vertical axis of the rotor is located in the upper left hand side of the picture. As it can be noticed, the maximum value of horizontal velocity vector component does not exceed 0.75 m/s. It can also can be observed, that the direction of fluid flow is opposite to the direction of rotor movement. The same manner is visible in the numerical results, which are shown in Figure 6. Also, the velocity values obtained numerically and experimentally are comparable. The difference between values of velocity obtained numerically and experimentally in the considered planes outside of the aerator comprises within the range of the measurement error. This proves that the numerical model of the single phase flow in the flotation cell is correct.

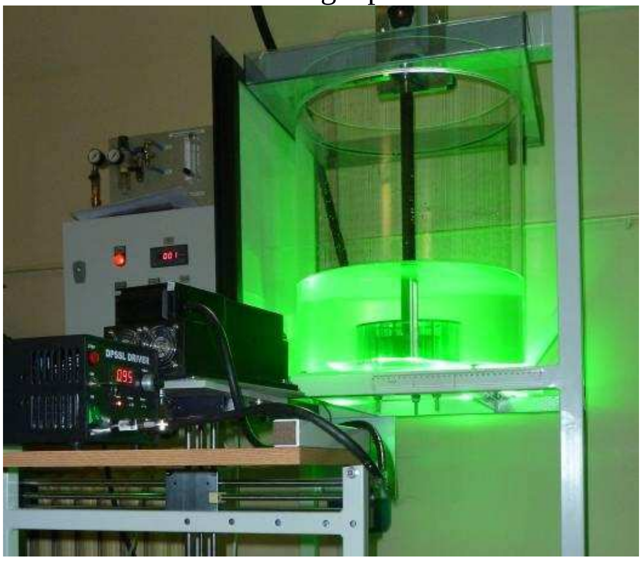

Figure 3. The laboratory stand.

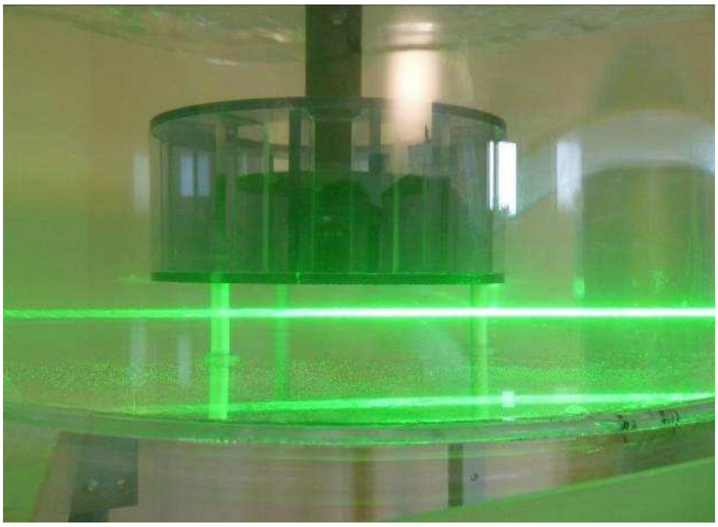

Figure 4. Measurement plane.

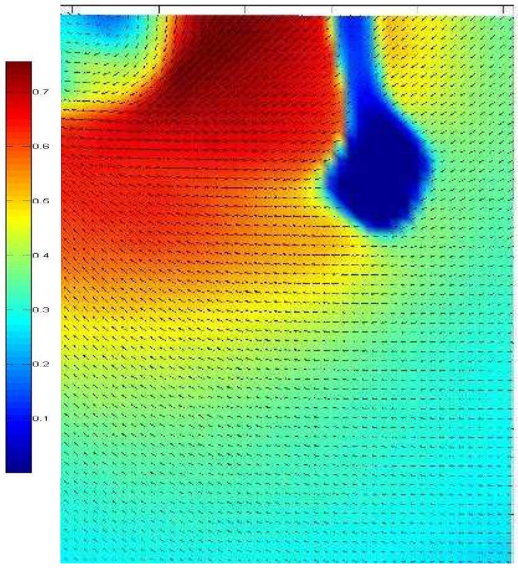

Figure 5. Distribution of velocity – PIV.

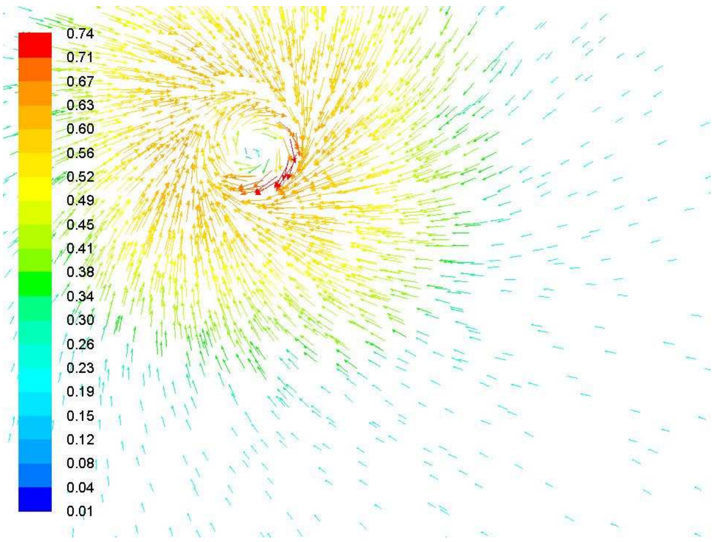

Figure 6. Distribution of velocity – numerical results.

4. NUMERICAL EXAMPLE

The numerical results presented in this section relate to a Denver type experimental flotation cell D12. Its aerator is shown in Figure 2. The goal of this investigation was to examine the possibility of using the Interfacial Area Concentration model to simulate the process of transport the water and air mixture, i.e. the two-phase flow in the flotation cell. As mentioned previously, ANSYS Fluent 13.0. commercial package was applied for this purpose. The computational domain and mesh are shown in Figure 7. The mesh consists of about 1 200 000 polyhedral elements. The MRF technique implemented in FLUENT has been employed due to existence of rotating and stationary elements in the model (rotor of the aerator and remaining elements). The steady state solution has been achieved as an asymptotic one, when solving an unsteady problem.

The following boundary conditions were prescribed:

- no slip wall boundary condition on the walls of the tank and aerator,
- rotation speed of the rotor 30 rad/s,
- free surface at the top of the tank,
- mass flow rate at the air supply.

As the initial condition, the fluid was assumed to be motionless.

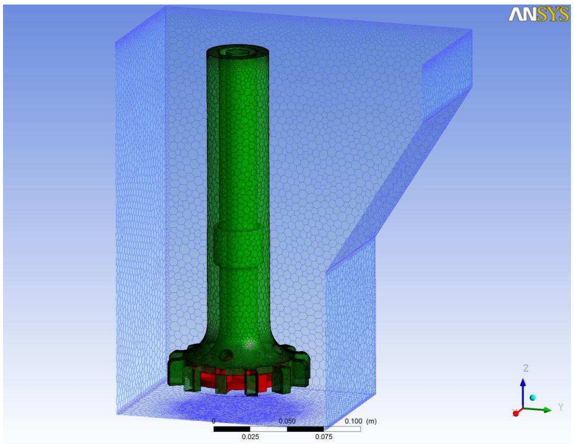

Figure7. Computational domain and polyhedral mesh.

Selected results are shown in Figures 8÷11. Distributions of the air volume fraction on selected surfaces are presented in Figures 8 and 10. They seem to be physically correct. The highest values of the air volume fraction were obtained in the aerator. The maximal values are located in positions observed in practise.

Velocity vectors of water in the domain of the aerator are shown in Figure 9. Their distribution is also in qualitative agreement with the one observed in the experimental flotation cell. The comparison of the interfacial area concentration and air volume fraction on the horizontal plane crossing the aerator is presented in Figures 10 and 11. Domains of higher values of the IAC are generally visible in domains of increased values of air volume fraction, which seems to be reasonable.

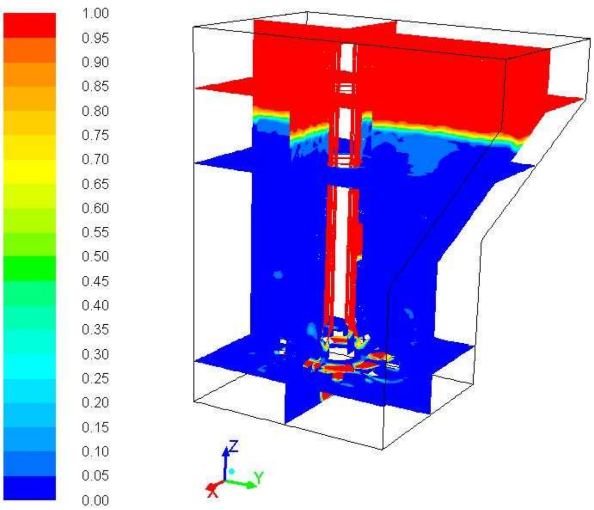

Figure 8. Air volume fraction on selected surfaces

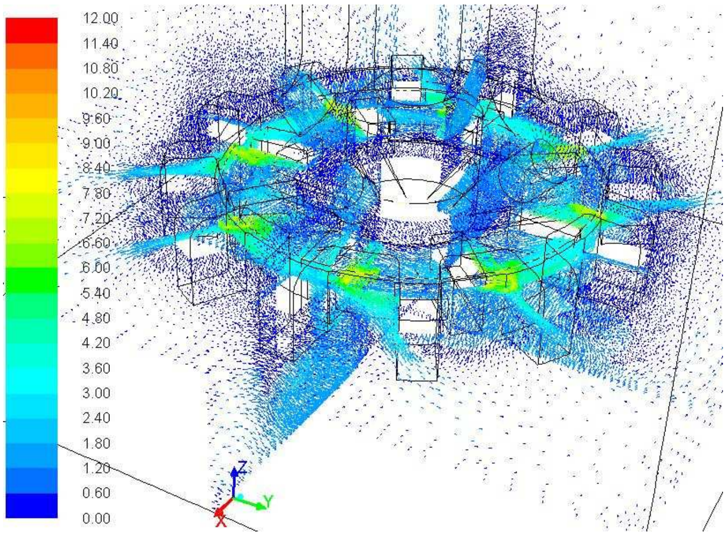

Figure 9. Velocities of water on the horizontal plane crossing the aerator

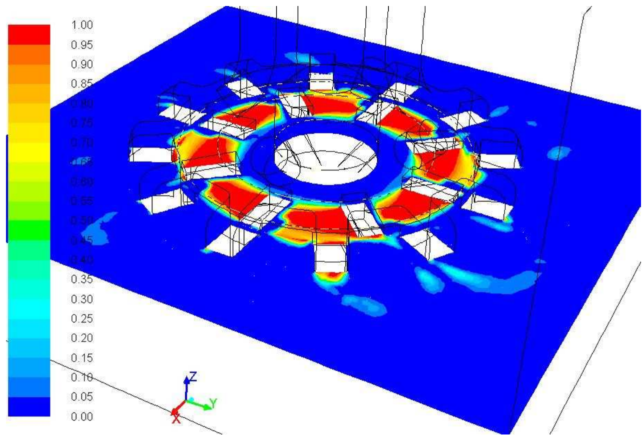

Figure 10. Volume fractions of air on the horizontal plane crossing the aerator

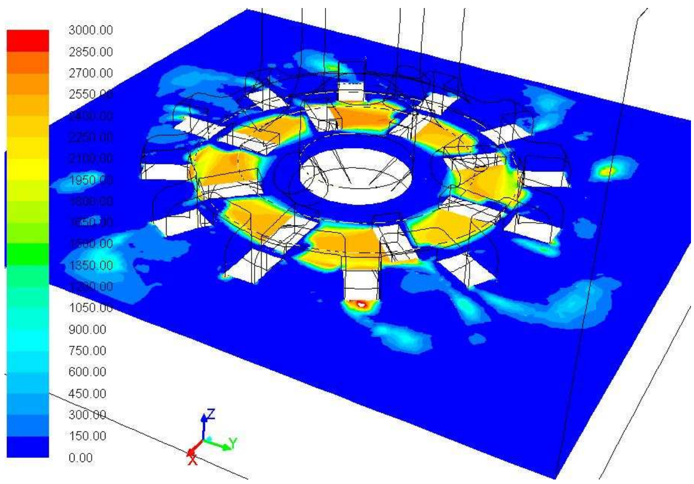

Figure 11. Interfacial area concentration on the horizontal plane crossing the aerator

5. FINAL REMARKS

The obtained results of modelling two phase flow of the water-air mixture in the experimental flotation cell D12 using ANSYS Fluent 13.0 package are satisfactory and they encourage further investigations. The results of simulations are generally in qualitative agreement with experimental observations. Satisfactory convergence has been achieved when solving the problem transiently to obtain an asymptotic stationary solution. The models used for the considered process require father validation. For this purpose, measurements will be carried out in the near future using the specially built experimental unit. The measurements worked out on this unit was employed to validate presented in the paper single phase model.

Acknowledgements

The financial support of The National Centre for Research and Development, Poland, within the grant NR02 0102-10 /2010 is gratefully acknowledged herewith.

REFERENCES

[1] *ANSYS-Fluent* 12.1.4; Theory Guide, April 2009.

[2] Deglon D.A., Meyer C.J., CFD modelling of stirred tanks: Numerical considerations. Minerals Engineering, 19, 2006, 1059-1068.

[3] Fic A., Sachajdak A., Szczygieł I., Mańka A.: Flow processes in the aerator of the flotation machine– preliminary simulations. XX Jubileuszowy Zjazd Termodynamików 2008. Termodynamika w Nauce i Gospodarce, Z. Gnutek, W. Gajewski (Eds.), Vol. I, 260-267, Oficyna Wydawnicza Politechniki Wrocławskiej, Wrocław 2008 (in Polish).

[4] Ishii M., Hibiki T., Thermo-Fluid Dynamics of Two-Phase Flow. Springer, New York 2010.

[5] Koh P.T.L., Schwarz M.P., CFD model of self-aerating flotation cell. Int. J. Mineral Process., 85, 2007, 16-24.

[6] Ramkrishna, D.: Population Balances: Theory and Applications to Particulate Systems in Engineering, Academic Press, 2000.

[7] Tiitinen J., Vaarno J., Grönstrand S., Numerical modelling of an Outokumpu flotation device. 3th Int. Conf. on CFD In The Minerals and Process Industries, Melbourne, Australia, 2003