19th Australasian Fluid Mechanics Conference
Melbourne, Australia
8-11 December 2014

A Review of CFD Modelling of Flotation Cells

A.R. Sarhan[1,2], **J. Naser**[2] **and G. Brooks**[2]

[1]Department of Engineering & Industrial Sciences
Swinburne University of Technology, Melbourne, Victoria 3122, Australia

[2]Department of Mechanical Engineering
University of Anbar, Ramadi, Anbar 31001, Iraq

Abstract

This paper presents an overview of the literature concentrated on using computational fluid dynamics (CFD) for modeling froth flotation process. The physics of flotation cells are described and the equations that were used in CFD models are reviewed. The utility of using the CFD model is illustrated in this work through presenting examples of research efforts for simulating froth cells. First, a CFD model which provides a realistic approach to flotation kinetic within flotation cell is introduced, and then the usage of population balance equation to track the bubbles break-up and bubbles coalescence is presented. An example of the use of CFD for predicting the failure of a thin liquid film within froth zone is also presented. The paper concludes by highlighting a number of important aspects not covered in open literature.

Introduction

Froth flotation is a complex physico-chemical process which is extensively used in mineral industry to separate hydrophobic minerals (valuable minerals) from hydrophilic ones (gangue). Recently, the usage of flotation process has been extended to a wide range of applications such as de-inking of recycled paper, water treatment and electrolyte cleaning (oil separation) [2, 13, 20]. Principles of flotation cell design and operations parameters have mainly been guided by experimental data and plant experience. This is because simulation techniques have previously been unable to deal with the study the complexity of the multiphase flow within the cells.

The last few decades have seen major advance in computer field which has allowed using computational fluid dynamics (CFD) methods to model the complexity of three-phase (air–water–solids) flows within the flotation cells. Computational fluid dynamics (CFD) has clearly emerged as a promising mean to predict the hydrodynamics and performance of these systems. This is because it has the potential to identify design and operational modifications to the flotation process, this has allowed increasing the understanding of complex flow patterns within the cell and mixing of gas bubbles with liquid and suspended particles, and provides an opportunity to analyse the performance of the flotation cell and thereby the recovery of the flotation cell without conducting real-time experiments [1, 12, 22].

In CFD modelling, a flotation cell is discretized into specific finite volumes where local values of flow fields are determined. The understanding of detailed flow physics that is gained by using this approach allows modification to existing equipment and operations to improve flotation performance. Furthermore, these results provide the opportunity of investigating bubble-particle interactions in a real environment [4, 6, 8].

This paper will discuss the use of CFD in flotation cell simulation. The physics of flotation cell is introduced and the equations that were used in CFD models are reviewed.

Flotation Cell Structure and Physics

A flotation cell could be generally divided into two distinct zones, as shown figure 1 [5], the pulp zone and the froth zone. The overall performance of the flotation system relies on the collective result of these two zones.

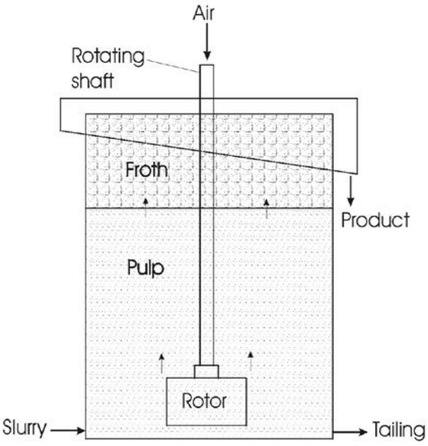

Figure 1. Principle of froth flotation [5]

During the flotation process, air is injected into suspension of ore and water where a small quantity of flotation reagent is frequently added. The surfactants modify the surface properties of ore particles as well as gangue particles, and it enhances the attachment of the particles to the bubbles. The bubbles collect the dispersed particles during their ascent through the pulp zone, they are then released in the froth zone where they are removed as a concentrate, where gangue particles are discard from the flotation cell as tailings [2, 13].

$$\frac{\partial(\alpha_i\rho_i)}{\partial t} + \nabla.(\alpha_i\rho_i U_i - \rho_i D_i \nabla\alpha_i) = S_i \qquad (1)$$

$$\begin{aligned}\frac{\partial(\alpha_i\rho_i U_i)}{\partial t} &+ \nabla.(\alpha_i\rho_i U_i \times U_i)\\ &= \nabla.\left(\alpha_i\left(\mu_{L,i} + \mu_{T,i}\right)(\nabla U_i + (U_i)^T\right)\\ &+ \alpha_i(B_i - \nabla P_i) + F_i + S_i U_i \qquad (2)\end{aligned}$$

The details of the above equations can be found in [9]. Recently, a population-balance technique has been employed to determine the local bubble size distribution within flotation cell [18, 19, 20]. It involves solving transport equations for the transfer of the number of bubbles by convection and turbulent diffusion, and changes in the bubble numbers by breakage and coalescence [9, 13]. There are several models available in the literature for bubble-breakage and coalescence that were used by researchers. By applying these models, it will allow for the bubbles to transfer between size groups during the breakage and coalescence processes. A general form of the bubble population balance equation can be written as follows:

$$\frac{\partial}{\partial t} n_i + \nabla.(n_i U_i) = B_B - D_B + B_C - D_c \qquad (3)$$

Where n_i is the number of ith bubble groups per unit volume, B_B and B_C are the birth rates of bubbles due to breakage and coalescence, DB and DC are the death rates of bubbles due to breakage and coalescence, respectively. For modelling flotation kinetics, the changing of the concentration of solid particles inside flotation cell is achieved by applying source and sink terms as in follows equation:

$$\frac{\partial(\alpha_i m_i)}{\partial t} + \nabla.(\alpha_i m_i U_i) = -S_a + S_d \qquad (4)$$

Where m_i is the concentration of particle, and S_a, S_d are sources or sinks specifying attachment and detachment rates, respectively [9].

Flotation Kinetics

The metallurgical process such as flotation process, which is very complex in terms of chemical reaction and interactions between phases, has adopted the CFD technique to understand the multiphase flow and associated phenomena in the system. The interactions between bubbles and solid particles and their motions in a flotation cell are the most important phenomenon governing a flotation process [12]. Currently, there are two major approaches to describe the flotation kinetics within flotation cell; both approaches are based on well- established models to predict flotation performance [3]. The first approach is a first-order process which is widely used in industry to illustrate the flotation kinetics with respect to the number of particles and bubbles [4, 6, 7]. It employs the residence time to predict the recovery of the floated particles, and the experimentally determined flotation rate constant. The second approach considers the recovery of the flotation cell as a combination of true flotation (the attachment of solid particles to gas bubbles) and entrainment (the suspension of solid particles in the water between bubbles in the froth). Although, the first order rate constant approach is insufficient to illustrate the mechanisms occurring in the froth phase, numerous researchers used it in their simulation to describe flotation kinetic within pulp phase [3]. The rate of removal of particles in a given volume represents the kinetic equation for the three sub-processes involving collision, attachment and detachment with respect to the number of density of particles and bubbles as follows [4, 6, 7]:

$$\frac{dN_{p1}}{dt} = -k_1 N_{p1} N_{bT}(1-\beta) + k_2 N_{bT}\beta \qquad (5)$$

Where N_{p1} is the number concentration per unit volume of particles available for attachment, N_{bT} is the number concentration of free bubbles, and k_1 and k_2 is the particle–bubble attachment rate constant and the particle–bubble detachment rate constant respectively. Generally, gas bubbles will have any number of particles attached as they move up toward the froth zone. The number of particles will change with time and position, as well as from bubble to bubble. The average loading parameter β in a given volume varies with time and position.

Modelling of Pulp Zone

A considerable amount of literature has been published on modelling of flotation cells. Only the pulp zone was considered in the majority of these studies and the effect of froth zone was neglected. Most investigations involve studying the complexity of three phase (gas–liquid–solids) flows within the cells [4, 6, 7, 9, 11, 12, 13]. However, all the studies treat geometry and hydrodynamics of flows inside flotation cell in a simplistic manner. First attempts at what made by different research groups are summarised below. A finite volume CFD framework has been used by [4] to build a three phase model of a flotation tank. The authors used a turbulent collision model to calculate the rate of bubble-particle encounters, employing the local turbulent velocity and the size and number concentrations of gas bubbles and particles obtained from CFD modelling. A constant bubble size of 1 mm in the cell was considered. The simulations were conducted on two computational domains representing a CSIRO flotation tank and a stirred tank with a Rushton turbine figure 2 & figure 3.

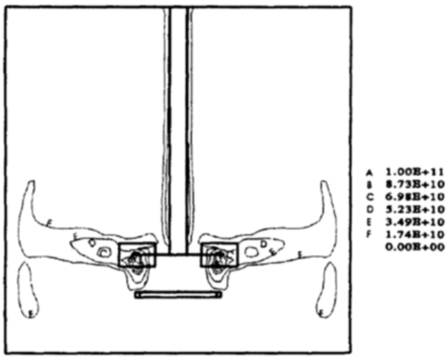

Figure 2. Contours of bubble-particle collision rate in Rushton turbine stirred tank [4].

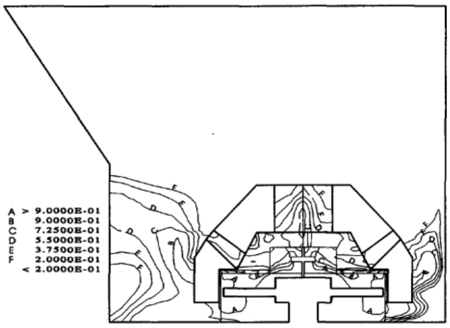

Figure 3. Contours of bubble-particle collision rate in CSIRO flotation cell [4].

The streamline impact of fine particles that are moving around a larger bubble was considered in the calculation of probability of collision. The collision frequency and the probabilities of collision, adhesion and stabilisation were employed to calculate the local attachment and detachment rates. The influences of the turbulence intensity on the local rates of attachment and detachment can be quantified, and thereby the local bubble loading can be estimated. The results have demonstrated that the flotation cells must be designed in way to ensure good mixing between the suspended solid particles and air bubbles in order to obtain high efficiency of the flotation process.

Koh and Schwarz [7] improved the model presented earlier. However, there is a contradiction in this work. The collision probability used in the model is valid for bubbles smaller than 1 mm whereas average bubble diameter used in the simulation was larger than this value depending on the speed of impeller. The paper shows that the bubble surface area flux is the limiting factor in the recovery rate at the pulp-froth interface, and also the rate of transport of bubble-particle aggregate to froth-pulp interface have a significant impact on flotation recovery. Koh and Schwarz [8] reported that the bubble coalescence rate is reduced at high concentration of frother, this leads to constant bubble size over the range of impeller speed. Koh and Smith [10] stated that there is an optimum stirring speed that produces a good agreement between attachment and detachment rates in the flotation cell.

Mirgaux et al. [13] proposed mathematical modeling and CFD simulation of molten aluminum purification by flotation in a Stirred cell in order to obtain on a better understanding of the physical phenomena that act in flotation cell and the optimization of the refining process. Three-phase system, namely, liquid aluminum, bubbles, and inclusions were considered in this work. The influences of agglomeration and flotation on the density number of inclusion were considered in this simulation, while the impact of fragmentation phenomena was not considered. However, bubbles are considered to be solid spheres with a constant and unique diameter that depends on the rotor speed and gas flow rate without any interaction between them. It was found that the turbulence properties (ε and k_t) reach their maximum value around the blades of the rotor where the shear is strong as shown figure 4.

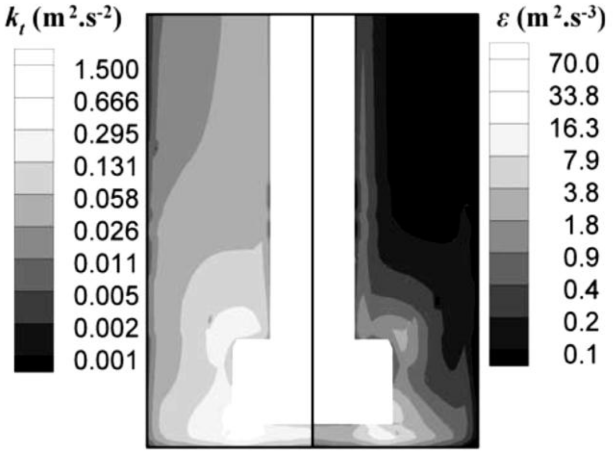

Figure 4. Kinetic energy k_t and its dissipation rate ε [13].

The results revealed that the efficiency of flotation process might be improved if we can obtain the zones of high turbulence intensity and the zones of high gas holdup to coincide.

Liu and Schwarz [12] performed computational fluid dynamics (CFD) to investigate the collision efficiency of bubble-particle with mobile bubble surface in turbulent flow. The influence of turbulence on small immersed particles was simulated using a stochastic model to produce a fluctuating flow field based on a steady RANS model, while the Lagrangian approach have been applied to calculate trajectories of particles. It was assumed that turbulent kinetic energy of eddies that are larger than bubble will effect on the bubbles motion, while turbulent kinetic energy of eddies that are smaller than bubble will effect on the particles motion. This work showed that turbulence affects bubble-particle collision in two ways: it effects the local average bubble relative velocity in the bubble-slurry and the turbulence dispersion of particles.

The froth phase is generally neglected in previous studies on performance of flotation cells. This represents a major deficiency in these studies because it is the froth phase that is primarily responsible for the improved metallurgical performance of flotation cells [21].

Modelling of Froth Zone

The pulp is only one part of the flotation process. It is physically linked to the froth, for which multiphase CFD models are being developed by a number of groups [14, 15, 16 & 23]. They contain a set of cases that were chosen two of them to introduce in this paper. The usage of CFD for predicting froth was illustrated in this work through the two examples presented. First, the mathematical model of solids motion within froth phase will be introduced, and then model for predicting the failure of a thin liquid film within froth will be presented.

Solids Motion within Froth Phase

The solids particles within froth phase (hydrophobic or hydrophilic) can be divided into two classes, attached particles to the bubble, which follows and moves with the bubbles, and the unattached material, which mainly follows the liquid and might move relative to the liquid by means of hindered settling and geometric and plateau border dispersion. Attached particles enter the froth attached to the bubble lamellae; however it can become unattached due to coalescence or bursting. On the other hand, unattached hydrophobic particles may attach to the bubble lamellae. The simulation is required in order to track the transfer of particles from being attached to being unattached which is considered as vital issue in the froth flotation. Neethling and Cilliers [16] presented a mathematical model for the motion of solids within flotation froth. The model involves the influence of the complex interacting phenomena observed in flotation froths, including the dispersion and settling of particles, the motion of the gas and liquid, and bubble coalescence and bursting. Figure five depicts an example of realistic results were, obtained by the simulation model which allows flotation performance trends and their magnitude to be explored under a variety of operating conditions.

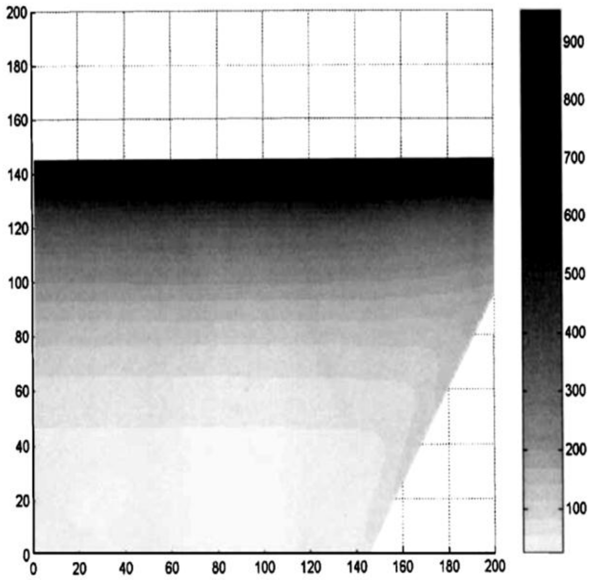

Figure 5. Concentration of desired mineral per volume off froth (kg solid=m^3 liquid) [16].

The Failure of a Thin Liquid Film

The interaction between a liquid–vapour (LV) interface and solid surface (s) is complex and has influence on a large range of applications; such as advanced oil recovery techniques and froth flotation. Numerous parameters affects the liquid–vapour interface such as the capillary pressure that acts on the liquid–vapour interface and the surface properties of the solid surface (contact angle). To date, the influence of particle behaviour on the film is poorly understood. However, some research recently revealed that the particles stabilise the film by increasing the capillary pressure required for this to occur [15]. Morris et al. [14] presented modeling results of the effect of non-uniform packing of spherical particles on the stability of thin liquid films. These results has been used to develop an approach to estimate the critical capillary pressure (P^*_{crit}) of a film required to rupture a particle stabilized thin film based upon the particle packing density and contact angle of the particles. In this study surface

Evolver [14], which is an interactive program for the modelling of liquid surfaces shaped by various forces and constraints, was used to simulate cells containing up to 20 particles, randomly packed in a thin liquid film. The results show P^*_{crit} proportional to a constant K and inversely proportional to the particle packing density, or area of film per particle A^*_{PP}. The value of K relies on the uniformity of packing of the particles; uniformly packed particles have a K of 4, while the randomly packed particles have a K of 2.31. Morris et al. [15] investigated the impact of the particle hydrophobicity and packing density which has on the structure and failure mode of a thin liquid film stabilised by a double layer of particles. It was found that a uniform double layer of particles in a hexagonal arrangement will stabilise a film to a much greater extent and over a larger range of contact angles than a single layer of particles in a hexagonal arrangement.

Conclusion

This paper discussed the usage of CFD in flotation cell simulation. The hydrodynamic environment inside flotation cell was introduced and the equations used in CFD models were reviewed.

The examples of using CFD model was illustrations in this work through presenting research efforts for simulating froth cells. The CFD model which provides a realistic approach to flotation kinetic within flotation cell, using population balance equation to track the bubbles break-up and bubbles coalescence was introduced. An example of the usage of CFD model for predicting the failure of a thin liquid film within froth zone also was presented.

It is concluded that the combination of pulp and froth models into a single simulation is desirable, but the differences in flow behaviour, and the complexity of mass and momentum exchange between pulp and froth phases presents significant challenges.

Acknowledgments

The authors would like to thank the Higher Committee for Education Development in Iraq (HCED) for their financial support.

References

[1] Chakraborty, D., Guha, M. & Banerjee, P.K., CFD Simulation on Influence of Superficial Gas Velocity, Column Size, Sparger Arrangement, and Taper Angle on Hydrodynamics of the Column Flotation Cell, *Chemical Engineering Communications*,**196**, 2009, 1102-1116.

[2] Ghaffari, A. & Karimi, M., Numerical Investigation on Multiphase Flow Simulation in a Centrifugal Flotation Cell, *International Journal of Coal Preparation and Utilization*,**32**, 2012, 120-129.

[3] Hu, W., Hadler, K., Neethling, S. J. & Karimi, M., Determining flotation circuit layout using genetic algorithms with pulp and froth models, *Chemical Engineering Science*,**102**, 2013, 32-41.

[4] Koh, P.T.L., Manickan, M. & Schwarz, M.P., simulation of bubble- particle collisions in mineral flotation cells, *Minerals Engineering*,**13**, 2000, 1455-1463.

[5] Koh, P.T.L., Schwarz, M.P., Zhu, Y. & Bourke, P., Development of CFD models of mineral flotation cells, *Conference on CFD in the Minerals and Process Industries*, CSIRO, Melbourne, Australia, 2003, 171-175.

[6] Koh, P.T.L. & Schwarz, M.P., CFD modelling of bubble–particle collision rates and efficiencies in a flotation cell, *Minerals Engineering*,**16**, 2003, 1055-1059.

[7] Koh, P.T.L. & Schwarz, M.P., CFD modelling of bubble–particle attachments in flotation cells, *Minerals Engineering*,**19**, 2006, 619-629.

[8] Koh, P.T.L. & Schwarz, M.P., CFD model of a self-aerating flotation cell, *International Journal of Mineral Processing*,**85**, 2007, 16-24.

[9] Koh, P.T.L. & Schwarz, M.P., Modelling attachment rates of multi-sized bubbles with particles in a flotation cell, *Minerals Engineering*,**21**, 2008, 989-993.

[10] Koh, P.T.L. & Smith, L.K., The effect of stirring speed and induction time on flotation, *Minerals Engineering*,**24**, 2011, 442-448.

[11] Liu, T.Y. & Schwarz, M.P., CFD-based modelling of bubble-particle collision efficiency with mobile bubble surface in a turbulent environment, *International Journal of Mineral Processing*,**90**, 2009, 45-55.

[12] Liu, T.Y. & Schwarz, M.P., CFD-based multiscale modelling of bubble–particle collision efficiency in a turbulent flotation cell, *Chemical Engineering Science*,**64**, 2009, 5287-5301.

[13] Mirgaux, O., Ablitzer, D., Waz, E. & Bellot, J.P., Mathematical Modeling and Computer Simulation of Molten Aluminum Purification by Flotation in Stirred Reactor, *Metallurgical and Materials Transactions*,**40**, 2009, 363-375.

[14] Morris, G., Neethling, S.J. & Cilliers, J.J., A model for the stability of films stabilized by randomly packed spherical particles, *Langmuir*,**27**, 2011, 11475-80.

[15] Morris, G., Neethling, S.J. & Cilliers, J.J., Predicting the failure of a thin liquid film loaded with spherical particles, *Langmuir*,**30**, 2014, 995-1003.

[16] Neethling, S.J. & Cilliers, J.J., Simulation of the effect of froth washing on flotation performance, *Chemical Engineering Science*,**56**, 2001, 6303-6311.

[17] Neethling, S.J. & Cilliers, J.J., Solids motion in flowing froths, *Chemical Engineering Science*,**57**, 2002, 607-615.

[18] Sattar, M.A., Naser, J., Brooks, G., Numerical simulation of creaming and foam formation in aerated liquid with population balance modeling. *Chemical Engineering* Science,**94**, 2013, 69-78.

[19] Sattar, M.A., Naser, J., Brooks, G., Numerical Simulation of Two-Phase Flow with Bubble Break-up and Coalescence Coupled with Population Balance Modeling. *Chemical Engineering and Processing: Process Intensification*,**70**, 2013, 66-76.

[20] Sattar, M.A., Naser, J., Brooks, G., Numerical Simulation of Slag Foaming in High Temperature Molten Metal with Population Balance Modeling. Procedia Engineering,**56**, 2013, 421–428.

[21] Tao, D., Luttrell, G.H. & Yoon, R.-H., A parametric study of froth stability and its effect on column flotation of fine particles, *International Journal of Mineral Processing*,**59**, 2000, 25-43.

[22] Wadnerkar, D., Utikar, R.P., Tade, M.O. & Pareek, V.K., CFD simulation of solid–liquid stirred tanks, *International Journal of Mineral Processing*,**23**, 2012, 445-453.

[23] Yang, R.Q., Wang, H.F. & Lui, J.C., Jet Flotation Column System Structure Design and Numerical Simulation of Two-Phase Flow, *Advanced Materials Research*, 2012, 655-661.