chemical engineering research and design 90 (2012) 2196–2201

Contents lists available at SciVerse ScienceDirect

Chemical Engineering Research and Design

journal homepage: www.elsevier.com/locate/cherd

Experimental studies and numerical model validation of overflowing 2D foam to test flotation cell crowder designs

K.E. Cole*, P.R. Brito-Parada, C. Xu[1], S.J. Neethling, J.J. Cilliers

Rio Tinto Centre for Advanced Mineral Recovery, Froth and Foam Research Group, Department of Earth Science and Engineering, Imperial College London, South Kensington Campus, London SW7 2AZ, United Kingdom

ABSTRACT

A computational fluid dynamics model of froth motion has been developed to assess different flotation cell designs. This work presents an implementation of the model in a 2D case, to compare the simulated bubble velocity distribution and streamlines to an experimental foaming system. The model uses finite elements to solve Laplace's equation for a potential function from which the foam velocity can be obtained. It requires the air recovery, or the amount of air that overflows a flotation cell as unburst bubbles, as an input parameter to calculate the foam velocity distribution and bubble streamlines. The air recovery was obtained by image analysis from a vertical, overflowing monolayer of foam (2D) created in a Hele-Shaw column, which mimicked important flowing properties of flotation froths such as coalescence. Inserts were included in the foam column to represent potential crowder designs for industrial flotation cells. Three different designs were chosen to compare the effect of insert depth and shape, including rectangles and a triangle. The effect of the insert design on the overflowing foam is obvious from visual assessment of the bubble streamlines and velocity distribution, which were closely agreed by both the experiment and model.

Keywords: Froth flotation; Flotation bubbles; Image analysis; Computational fluid dynamics; Process optimisation

1. Introduction

Flotation machine design is constantly evolving to meet specific requirements of a particular industrial plant. The recent trend has been to increase the size of the flotation cell, which increases the froth volume. If this froth can only overflow at the vessel lip, the centre of the vessel becomes a large stagnant zone of froth which does not report to the concentrate (Zheng et al., 2004; Zheng and Knopjes, 2004). Therefore two different types of devices have been placed in the froth zone to improve mineral recovery: launders to increase the surface area for overflowing froth, and crowders to direct the froth flow. In this work, a model of foam flow has been developed that can predict the effect of insert design on overflowing foam, to give insight into the effect of crowder design on flotation performance.

The function of a crowder is to decrease the cross sectional area at the top of the froth to improve the froth removal dynamics in the flotation cell. The walls of a crowder provide a surface to direct froth toward the overflow launder. They reduce the amount of air required for operation, or alternatively increase the volume of overflowing froth for a particular air rate. Crowders usually extend from the impeller outwards and from the outer wall inwards, although they can be placed mid-cell directing towards the weir. A crowder design from a flotation cell at Rio Tinto's Northparkes mine is shown in Fig. 1, where the angled surface of the crowder directs the froth to overflow the weir on the right side of the image. Degner (1997) patented a crowder device designed to improve the removal of froth, reducing the amount of air needed to produce froth and thus the energy needed to power the cell rotor. Fuerstenau et al. (2007) described the effect of adding a conical crowder above the impeller hood to a Wemco 1 + 1 flotation machine, which increased the copper recovery by 18%. For a reasonably straight forward engineering solution, crowders can have a significant effect on flotation efficiency.

Computational fluid dynamics (CFD) is a powerful tool that can be exploited to optimise the performance of existing

---

* *Corresponding author*. Tel.: +44 20 75947145; fax: +44 20 7594 7444.
E-mail address: k.cole07@imperial.ac.uk (K.E. Cole).

Received 4 January 2012; Received in revised form 12 April 2012; Accepted 11 May 2012

[1] Present address: Institute of Computer Science and Technology, Peking University, China.

0263-8762/$ – see front matter © 2012 The Institution of Chemical Engineers. Published by Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.cherd.2012.05.009

Fig. 1 – Photograph of a crowder device from a flotation cell at Rio Tinto's Northparkes mine.

systems and to develop new technology to maximise process efficiency. The flow of flotation froths can be modelled as a potential flow. Numerical models for flowing foams based on solving Laplace's equation for a stream function (Moys, 1984; Murphy et al., 1996; Neethling and Cilliers, 1999) have proven useful but present some limitations; in particular the fact that they are restricted to two-dimensional simulations.

Brito-Parada et al. (2012) developed a numerical model for the trajectory and velocity of a flowing foam that uses finite elements on unstructured meshes to solve Laplace's equation for a scalar function. The selection of a potential function over the stream function allowed the implementation of the model not only for two dimensions but, for the first time, also for three-dimensional cases.

As CFD models require the mathematical descriptions of a process to represent the phenomena as accurately as possible, the simulations must be validated with experimental data. This presents a problem for flotation modelling, as flotation froths are opaque and fragile meaning there is a lack of experimental methods to measure their internal behaviour. Cole (2010) developed an overflowing foam column to measure the internal bubble properties optically. Image analysis measurements were possible as the Hele-Shaw column contained a monolayer of bubbles, or quasi 2D foam. The two phase chemical system mimicked a flotation froth as it was designed to provide a dynamic, viscous and coalescing foam that overflowed and burst at the foam surface. Measurements of the bubble size distribution in this 2D column were used to validate a coalescence model in Tong et al. (2011), which combined a liquid drainage model and a population balance model to predict the bubble size distribution over the height of the foam.

Image analysis methods can also be used to measure the bubble velocity and average flow streamlines in the 2D column, presenting a useful opportunity to validate the model of foam flow developed by Brito-Parada et al. (2012). The overflowing foam behaviour can be quantified in terms of the air recovery, or amount of air that overflows as unburst bubbles. This is important for the foam flow model which cannot yet predict the air recovery, making it a key parameter in determining the bubble velocity profile. As recent studies have highlighted a link between the air recovery and performance of a flotation cell (Ventura-Medina et al., 2003; Barbian et al., 2007), the inclusion of air recovery in this analysis allows direct comparison to a flotation cell.

2. Model for froth motion

By considering irrotationality and incompressibility of the foam, both valid assumptions as discussed by Neethling and Cilliers (2003), Laplace's equation is solved in the model by Brito-Parada et al. (2012) for the potential field ϕ:

$$\nabla^2\phi = 0. \tag{1}$$

The boundary conditions for the model are based on the geometry of the flotation cell, the air flowrate to the tank Q_a, and also on the air recovery α, an important variable in froth flotation. Assuming that the flux through the boundaries is uniform:

- At the solid walls:

$$\nabla\phi \cdot \mathbf{n} = 0. \tag{2}$$

- At the liquid/foam interface, A_I:

$$\nabla\phi \cdot \mathbf{n} = -\frac{Q_a}{A_I}. \tag{3}$$

- At the outflow, A_O:

$$\nabla\phi \cdot \mathbf{n} = \frac{Q_a\alpha}{A_O}. \tag{4}$$

- At the top surface of the foam, A_S:

$$\nabla\phi \cdot \mathbf{n} = \frac{Q_a(1-\alpha)}{A_S}. \tag{5}$$

As can be seen from the boundary conditions, air recovery is an important input for the model. Air recovery has been demonstrated to play a key role on the performance of flotation systems (Barbian et al., 2007), and it has been found not only that a peak in air recovery exists in industrial flotation froths but also corresponds to the air rate at which the highest overall mineral recovery is obtained (Hadler and Cilliers, 2009).

The numerical model for foam velocity of Brito-Parada et al. (2012) was implemented in Fluidity (AMCG, 2011), the finite element code which provides the computational framework for the model. In particular, the use of adaptive anisotropic meshes provides a means of capturing strong gradients of the flow and therefore is ideal for highly detailed simulations.

Experiments of flowing foams with inserts at the top, resembling crowders in flotation cells, can be used to study the effect these have on the flow. They also provide case studies to assess the capability of the numerical model to predict foam flow patterns and the velocity distribution of the flowing foam.

3. Experimental method

A Hele-Shaw type column is composed of two vertical parallel plates, with a narrow space in between. In this work, a plate separation of 5 mm was used to create a foam with only one layer of bubbles between two Perspex plates of size 400 mm × 400 mm. Two internal weirs with outwardly angled tops created a foaming space and an area for the liquid from the overflowing foam to recycle, as shown in Fig. 2. Air was supplied at the base of the column at a rate of 2.5 l/min.