Fifth International Conference on CFD in the Process Industries
CSIRO, Melbourne, Australia
13-15 December 2006

UNDERSTANDING FROTH BEHAVIOUR WITH CFD

Jan CILLIERS

Royal School of Mines, Imperial College London, UNITED KINGDOM

ABSTRACT

This paper will discuss the use of CFD in foam and froth modelling. The physics of froths is introduced and the methodology of combining the models for each phase into a complete description is reviewed.

The utility of the CFD model is illustrated in two ways: First the equations are solved explicitly, giving a clear interpretation of process variables and their importance. Second, an example of the use of froth CFD for equipment design is given. Both cases are compared with experimental data.

The paper concludes by highlighting a number of important issues that remain to be addressed.

INTRODUCTION

Froth flotation is a particle separation process used for collecting selectively the small fraction of valuable mineral from a mixed ore. Froth flotation is based on differences in particle hydrophobicity and is the largest tonnage separation process worldwide. In flotation, the particles, after being finely ground and suspended in water (the pulp), are treated with a surfactant to make the sulphur-bearing minerals hydrophobic. Air is blown into the pulp mixture, and the particles attach to the bubbles, rise to the surface and are collected continuously in an overflowing froth (the concentrate). Modern flotation tanks have a volume of $100m^3$ to $300m^3$, and plants have 3 or 4 flotation stages, each with 6 to 10 tanks.

Although occupying less than 10% of the flotation cell volume, the froth behaviour largely determines the fractional and relative recoveries of the valuable and waste minerals from the pulp to the concentrate. Design issues have included froth flow modification and addition of water to enhance the separation. To date, operating and design variables have been changed largely by empirical observation and trial-and-error. This is not cost-effective and the potential for a CFD-model approach to froth flotation design and optimisation is significant. However, froths have unique properties that make their modelling difficult and which must be considered.

This paper will describe first foam structure and the physics of flotation froths. The significant structural changes in the froth will be detailed. Models for the motion of the bubbles, the liquid and the solids will be introduced, and how they are combined to give a CFD-type description.

The utility of the approach will be illustrated by extracting an explicit solution for the overflowing water rate that allows interpretation of observed industrial behaviour. Full CFD simulations will compare surface and internal wash-water distribution.

The paper will also discuss the key outstanding issues in froth modelling, and the potential of implementing these.

FOAM STRUCTURE AND PHYSICS

The Structural Components: Lamellae, Plateau borders and Vertices

Foams and froths structure is well-defined by the physics of minimal surfaces (Weaire and Huzler, 1999). Consider a vertical cross-section through a typical two-phase foam. The foam is formed by bubbles freely rising through the liquid until they meet the foam-liquid interface (Figure 1).

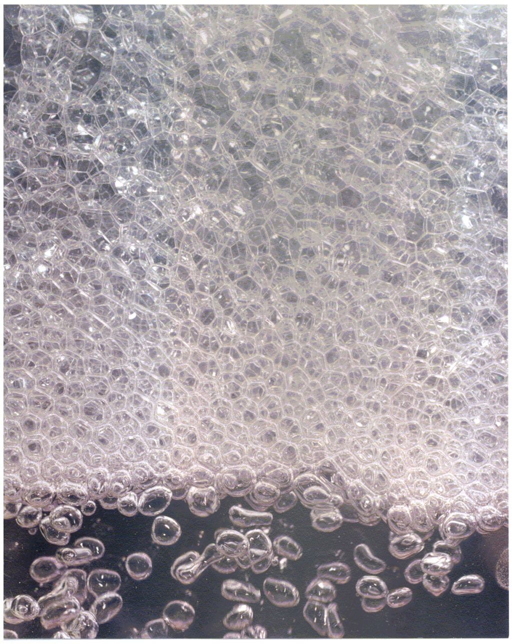


Figure 1: Vertical cross-section through a typical flowing foam showing rapid decrease in liquid content and bubble growth

In the lowest part of the foam, the bubbles are round and essentially a collection of close-packed spheres. The liquid content very rapidly drops above the interface and within a few bubble diameters the foam is significantly drier, where the bubbles take on an angular shape. These polyhedral bubbles coalesce when the lamella separating two bubbles fails.

Plateau borders are channels formed where three lamellae meet. Because of surface tension, the angle between three lamellae forming a Plateau border is always 120º, and only three lamellae form a stable Plateau border. Plateau borders have a well defined shape, with a curvature determined by the liquid content and bubble size. Figure 2 shows the shape of a Plateau border. Four Plateau borders meet in a vertex, at the tetrahedral angle. Figure 2 shows a micrograph of a solidified liquid foam and for comparison the simulated shape of a vertex, based on minimum surface area calculations.

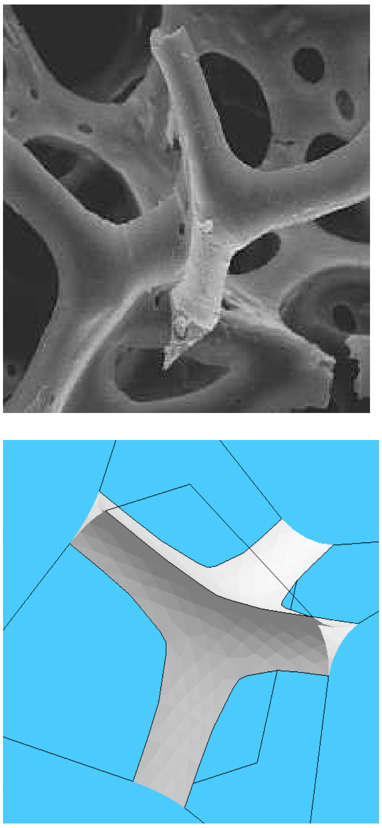


Figure 2: Plateau borders in a solidified foam (top) and a simulated vertex (below)

The Plateau borders and vertices form an interconnected network of channels through the foam. The network of Plateau borders and vertices contains virtually all the liquid in a foam. The gas-liquid interface curvature exerts a negative pressure on the lamellae and draws liquid into Plateau borders and vertices. Flotation froths should therefore be regarded as a network of liquid channels, the dimensions of which vary with liquid content and bubble size. The bubble size determines the Plateau border length per volume, λ, and is inversely proportional to the square of the bubble radius, *R*.

$$\lambda \approx 1.71/R^2 \qquad (1)$$

Bubble Coalescence and Bursting

Flotation froths are unstable and the bubbles coalesce in the froth and burst on the surface. Prediction of coalescence and bursting is highly complex, and only limited success has been had in elucidating relationships between, for example, the loading of particles on the lamellae and the fraction of bubbles bursting.

The froth rapidly undergoes tremendous structural change. Consider a typical froth, 100mm thick, being formed continuously from a superficial gas velocity of 20mm/sec giving an average bubble lifetime of 5 seconds in the froth. If a 0.5mm bubble entering coalesces to 16mm on the surface, it has to coalesce 15 times while passing through the froth; three coalescence events every second. The Plateau border length per volume reduces by a factor of 900, while 96.8% of lamella surface area is lost by coalescence. These changes have a significant impact on the separation.

Surface bursting is quantified by air recovery, α, the fraction of air entering the froth that overflows the weir, i.e. as unburst bubbles. The air recovery is, in general, surprisingly low, and values below 20%, and as low as 5% are commonly observed. Prediction of bubble coalescence and surface bursting remain one of the most challenging aspects of modelling froth behaviour.

The question is how to construct a CFD-type model for a system that has such a wide range of scales, phases and rapidly undergoes large changes. The emphasis here will be on the bulk foam motion and that of the liquid, as this, to a great extent, determines the behaviour of the solids.

A FLOWING FOAM CFD MODEL

The Bulk Foam

The time average motion of the bubbles in a flotation froth approximates flow of a continuous phase through a relatively simple geometry and can be described using the Laplace equation (Murphy et al., 1996). The fraction of air entering the froth that bursts on the surface defines the upper boundary condition and determines the flow trajectories.

The Liquid and the Drainage Equation

The motion of liquid and the local liquid content affect significantly the overall separation performance. Liquid draining through foams and froths shows flow behaviour that is markedly different from that of liquid draining through, for example, a packed bed of solid particles. For a particular bubble size, the Plateau border area changes as the liquid content changes. In packed beds the channel diameters are fixed.

Flotation froth has a low volumetric liquid fraction, allowing use of the foam drainage equation (Verbist et al., 1996, Leonard and Lemlich, 1965). This is obtained from a liquid force balance in Plateau borders; gravity downwards, viscous drag as the liquid moves relative to the gas bubbles and capillary suction brought about by changes in the Plateau border interface curvature as the Plateau border area, A, changes.

The drainage equation for low liquid flowing foams is:

$$v_l = -k_1 A - \frac{k_2}{\sqrt{A}} \frac{dA}{dy} + v_g \qquad (2)$$

$$k_1 = \frac{\rho g}{3 C_{PB} \mu} \quad \textit{and} \quad k_2 = \frac{\left(\sqrt{\sqrt{3} - \frac{\pi}{2}}\right)\gamma}{6 C_{PB} \mu}$$

In equation 2, v_l is the velocity of the liquid (or slurry in the case of flotation froths), A is the cross-sectional area of the Plateau border, and v_g is the upward velocity of the gas. In the drainage equations ρ, μ and γ are the density, viscosity and surface tension of the slurry in the Plateau borders. For flotation froths with immobile, solid-coated interfaces, $C_{pb} \approx 49$. The k_1 term balances gravity and viscosity, and k_2 balances capillary suction and viscosity.

The drainage equation is transformed to 2-D, combined with a liquid continuity equation and solved for the local Plateau border area (Neethling et al., 2000). The local liquid content is calculated by multiplying with the Plateau border length per volume for the local bubble size.

Solids Motion

All the particles in the froth, be they hydrophobic or hydrophilic, must be considered as one of two types, those particles attached to the bubble lamellae, and those particles that are unattached and free to move through the Plateau borders. The attached particles enter the froth attached to the bubble lamellae, and move on the same trajectory as described by Laplace's equation.

The unattached particles move freely through the Plateau border network with the liquid, but also relative to the liquid due to dispersion and gravity. Attached particles detach from bubble lamellae and become unattached due to surface bursting or coalescence in the froth; bubble surface area reduces particles on the lamellae transfer to the Plateau borders. The transfer of particles from being attached to being unattached is a key process in the froth and requires the simulator to track and label particles.

This paper will not dwell further on the modelling of the particle motion; a more complete description has been published elsewhere (Neethling and Cilliers, 2002b).

The Overall CFD model

The equations that describe the liquid are iteratively solved on a finite volume grid. The boundary conditions are well-defined by the physical system limits.

The following section will describe two examples of froth simulations that have clarified aspects of flotation froth behaviour. It will be shown that the construction of a physics-based CFD model allows the underlying mechanism that produces a process observation to be elucidated.

THE CFD MODEL APPLIED

The mathematical description of all the phases in a flotation froth, using equations that describe accurately the froth structure and physics, gives two important advantages over empirical equations. First, the role of each variable is explicit, and its relative importance is immediately clear from the equation form. Interactions between variables are clear, and the confounding of variables commonly found when experimenting is eliminated.
Second, only a mathematical description of this type is satisfactory when design modifications are required. The boundary conditions represent real dimensions and values, and can be manipulated.

This will be illustrated first by developing equations to predict explicitly the water recovery from an overflowing froth, and then by comparing two wash-water distribution designs.

Physical interpretation: Froth Depth and the Overflowing Water Rate

The foam drainage equations can be solved explicitly to predict the liquid flowrate to the concentrate. This is useful as it has been shown both experimentally (Engelbrecht and Woodburn, 1975) and theoretically (Neethling and Cilliers, 2002a) that the unwanted solids rate to the concentrate follows closely the water rate.

The liquid drainage equations are combined with relationships between the bubble size overflowing the weir (d_{bubble}) and Plateau border dimensions and solved using suitable boundary conditions. The solution for a typically low air recovery (α<0.5) gives the water rate to concentrate:

$$\text{Water Rate to Concentrate} \propto A_{Column} \frac{v_g^{\,2}}{d_{bubble}^{\,2}} (1-\alpha)\alpha$$

(3)

Equation 3 (Neethling et al, 2003) includes important physical system properties; the column area, the gas velocity, the bubble size and the bursting rate. However, an important operating variable conspicuous by its absence is the froth depth. It is a commonly held belief that deeper froths have a lower water overflow rate because there is "more drainage". While this is generally observed, the reason for it happening is incorrect.

The capillary suction term in the drainage equation rapidly reduces above the liquid-foam interface, and hence the Plateau border area in the froth is reasonably constant between the froth surface and a few bubbles from the pulp-froth interface. The "drainage" effect is therefore small.

Experiments show (figure 3, from Neethling et al, 2003) that for a non-coalescing, overflowing foam the liquid overflow rate is independent of foam depth.

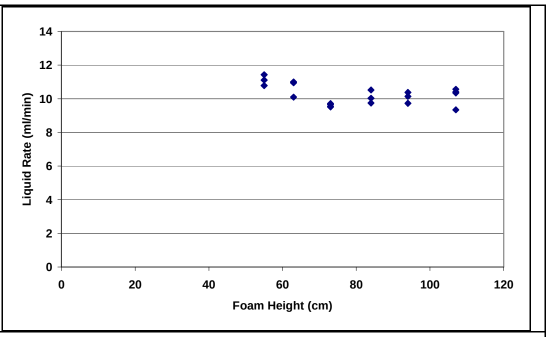


Figure 3: The effect of foam height on overflowing liquid rate for a non-coalescing foam, showing that the change in height does not affect the liquid rate.

The reason for the observed decrease in liquid rate with froth depth is clear from the physics, in equation 3. The Plateau border area per volume reduces as the square of the bubble diameter (from equation 1) and the change has only to be very small to give a significantly lower water rate. Further, the air recovery will decrease as more bubbles burst. The reduction in the overflowing water rate observed when the froth depth increases can therefore be attributed to increased coalescence and bursting, both as a result of the increased residence time.

An important outcome of explicitly solving the CFD model for the overflowing water rate is to identify key process variables that must be controlled and measured. In equation 3, the effect of air rate is squared. In industrial practise, the air rate is frequently a control variable for the solid flowrate. However, the strong effect on the water rate confounds its role and makes control difficult.

The overflowing bubble size effect is also squared, and affirms the importance of accurate froth surface bubble size measurement. Current systems give a reliable measure of the surface lamellae area distribution; this is not the same as the bubble size and must be taken into account. Further, changes in air rate will affect the incoming bubble size and also the size overflowing. An increase in bubble size reduces significantly the lamella surface area, and hence the behaviour of the valuable solids. The solids and liquid are both affected by the coalescence, but in different ways. Knowledge of the relative magnitude and trends allow process manipulation and optimisation.

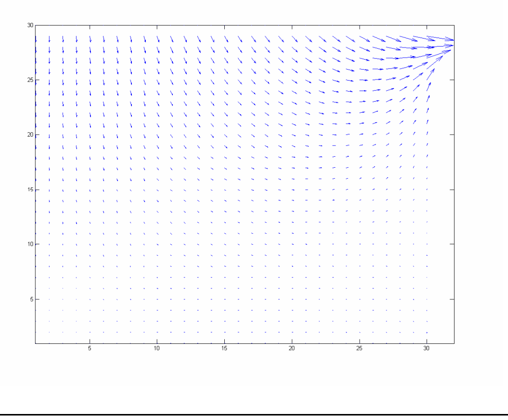
Finally, the effect of air recovery (or bursting rate) is non-linear. Air recovery is not measured routinely. The CFD clearly shows that it should be, if the concentrate flowrate is to be controlled.

Equipment Design: Wash water addition

The use of wash-water in froth columns is widespread. Water is added to create a downward liquid flow through the froth, and to wash out unwanted, entrained particles.

In terms of physical design, there are two options on how to add the wash-water; either on top of the froth or below the froth surface. Internal froth washing has gained favour as it is considered to not produce a wetter concentrate or reduce the recovery of valuable mineral to the same extent as surface washing. There are no clear guidelines for which addition method is suitable under which conditions.

This can be investigated using the CFD foam model, as the wash-water rate and addition point are simply boundary conditions of the simulation. It will not be attempted here to give a conclusive answer to whether internal or surface froth washing should be used, as there are further factors such as distribution patterns, water rates and cost. Instead, the motion of a uniform distribution of wash-water across the froth surface and internally will be illustrated, and compared with industrial data. Figures 4 show the simulation results for wash-water motion from the addition point through the froth depth.



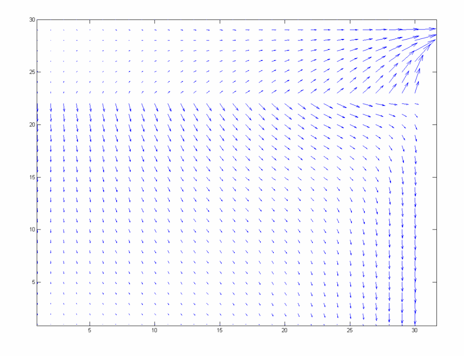


Figure 4: Liquid flow velocity vectors for evenly distributed froth washing from the froth surface (top) or inside the froth (bottom).

In these simulations, the froth is moving upwards and overflowing at the top right. Figure 4 (top, surface water addition) shows that there is a strong liquid flux that directly overflows without first entering the froth and washing it. This has two effects; first the concentrate solids concentration is reduced and second the effective amount of water entering the froth is reduced. Both result in poorer operation. Figure 4 (bottom) clearly shows the wash water addition and the flow downward flow pattern.

Figure 5 shows the froth relative liquid fraction for surface addition (top) and internal addition (bottom). Both additions show a gradual decrease towards the upper surface as a result of bubble size changes. For surface addition, the overflowing froth has a higher liquid content.

It must be noted that recent theoretical results have indicated that even small water additions will result in a churning motion in the froth, a so-called convective roll (Neethling et al., 2005). These do not appear in the simulations.

Figures 6 and 7 below show industrial operation results for surface and internal froth washing. Note that the aim of the wash water addition is to reduce the movement of fine undesired particles (here referred to as “insolubles”) into the concentrate.

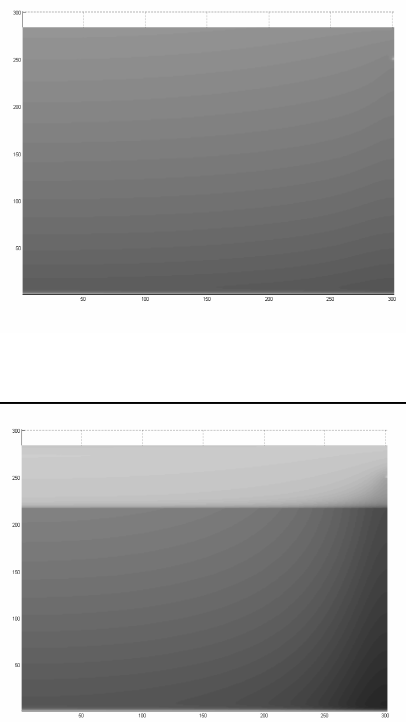


Figure 5: The froth relative liquid fraction for surface (top) and internal (bottom) water addition.

Figure 6 shows that the total solids fraction in the concentrate is consistently lower when the froth is washed from the surface than when washed internally under the same conditions. This is in agreement with the simulated flow trajectories in figures 4 and the froth liquid content in figure 5.

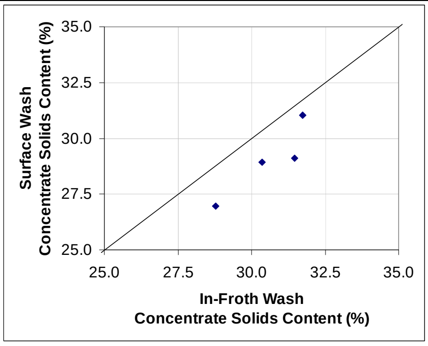


Figure 6: Comparison of solids fraction in concentrate for surface and internal froth washing, showing that internally washed froths are drier under equivalent conditions

Figure 7 shows the effect on one aspect of the separation performance. An increase in the wash-water rate leads to the desired decrease in insolubles recovery. More importantly, the recovery of insolubles is lower for in-froth washing than surface washing, at the same water rate, i.e. the separation performance is improved. These results are in complete agreement with the simulation predictions.

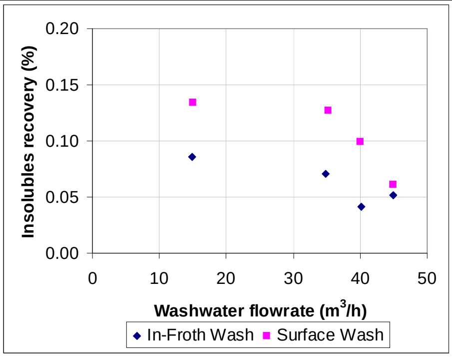


Figure 7: Insolubles recovery for surface and in-froth washing showing that at equivalent water rates in-froth washing is superior.

KEY PROBLEMS REMAINING

From the above examples it would appear that the modelling of flowing foams and flotation froths has come to the point where it can be confidently applied without further theoretical development. This is not the case. While there has been considerable success which has led to significant process changes and improvements in flotation performance, there are a number of significant modelling and experimental issues remaining.

Bubble coalescence and surface bursting

The surface bursting rate and the bubble size change inside the froth both are determined by lamella failure; either between two bubbles, or between a bubble on the surface and the atmosphere. In the two examples given, the bubble size remained almost constant throughout the froth depth and empirical models have been used to model the bubble size profile required in the models. Similarly, the air recovery, α, determined by the rate of failure of lamellae on the froth surface, was assumed known. This is an important boundary condition for the foam flow trajectory and the liquid rate to the concentrate, but cannot be predicted *a priori* at present.

Theoretical equations have been developed that link the probability of lamella failure to the predominant disturbances and modes of failure in the froth (Neethling and Cilliers, 2003). These have not been rigorously tested, although the probabilistic approach successfully predicts dynamic foam height from micro-scale two-bubble coalescence data (Cilliers *et al*., 2002). Recent work by Neethling et al. (2005) has shown that fundamental parameters for film failure can be estimated from foam growth and collapse rates. Whether the film failure probability on the surface and in the froth is the same remains an open question.

Developing and verifying models for determining from first principles the surface film failure and bubble coalescence remains one of the most challenging aspects of understanding in detail the froth behaviour.

Particle behaviour

This paper has focused on the liquid in the froth. The solids complicate matters significantly, and suffice it to say that there are also complex questions regarding the behaviour of particles in the froth. As noted above, the stabilising role of particles in lamellae is well-known (e.g. Kam and Rossen, 1999). However, the effects of particle size, shape and hydrophibicity are less well understood and, due to the complex interaction of their motion with the froth structure, difficult to incorporate into the models. This is undoubtedly a most important and difficult problem.

Further issues are the re-attachment of detached particles from the Plateau borders onto the lamella (Ata et al, 2002), and, related to this, the possible change in lamellae grade due to coalescence.

These problems raise important issues about multi-dimensional particle size, density and hydrophobicity distributions and how they are accommodated satisfactorily in CFD models, not only in those for froths.

Combined pulp-froth models

The froth is only one part of the flotation process. It is physically linked to the pulp, for which multi-phase CFD models are being developed by a number of groups (.

Linking the upper boundary of a 3-phase CFD model of a turbulent, mixed system to the lower boundary of a slow-flowing foam system is required. In particular, the interface between the pulp and the froth where mass and momentum transfer occurs remains unexplored.

CONCLUSIONS

This paper has discussed the use of CFD in foam and froth modelling. The physics of froths was introduced and the methodology of combining the models for each phase into a complete description reviewed.

The utility of the CFD model was illustrated in two ways: First the equations were solved explicitly, giving a clear interpretation of process variables and their importance. This allowed physical interpretation of observed industrial behaviour, and indicated potential control improvement routes.
Second, an example of the use of froth CFD for equipment design was given. This compared internal and surface wash water addition, and showed the reason for the industrial observations.

Both cases were compared with experimental data, and shown to predict the correct behaviour and trends.

The paper concluded by highlighting a number of important issues that remain to be addressed. In particular, the failure of bubble films leading to coalescence in the froth and bursting on the surface were highlighted.

Finally, the combination of pulp and froth models into a single simulation is desirable, but the differences in turbulence and flow behaviour, and the complex mass transfer across the interface are significant challenges.

REFERENCES

ATA, S., AHMED, N. and JAMESON, G..J., (2002), "Collection of hydrophobic particles in the froth phase", *International Journal of Mineral Processing*, **64**, 101-122.

CILLIERS, J.J., NEETHLING, S.J., SPYRIDOPOULOS, M.T. and SIMONS, S.J.R., (2002), "The Failure of Thin Films Between Bubbles – from micromanipulation to foam stability", *Proceedings, Flotation and Flocculation: From Fundamentals to Applications*, (Ed: J Ralston, J. Miller and J. Rubio), Strategic Conference and Workshop, Snap Printing, Australia, 169-17

ENGELBRECHT J.A. and WOODBURN E.T., (1975). "The effect of froth height, aeration rate and gas precipitation on flotation", *Journal of South African Inst of Mining and Metallurgy*, **10**, 125-132

KAM, S.I and ROSSEN, W.R. (1999), "Anomalous Capillary Pressure, Stress and Stability of Solids-Coated Bubbles". *Journal of Colloid and Interface Science*, **213**, 329-339.

LEONARD, R. and LEMLICH, R., (1965), "Laminar longitudinal flow between close-packed cylinders", *J. Chem Eng. Sci.* **20**, 790-791

MURPHY, D.G., ZIMMERMAN, W. and WOODBURN, E.T., (1996), "Kinematic model of bubble motion in a flowing froth", *Powder Technology*, **87**, 3-12

NEETHLING, S.J. and CILLIERS, J.J., (2002a), "Entrainment of solids into flotation froths", *International J of Minerals Processing*. **64**, 123-134

NEETHLING, S.J. and CILLIERS, J.J., (2002b), "Solid motion in foams" *Chem Eng Sci*. 57, 607-615

NEETHLING, S.J., CILLIERS, J.J. and E.T. WOODBURN, (2000), "Prediction of the water distribution in a flowing foam", *Chem Eng Sci*, **55**, 4021-4028.

NEETHLING, S.J., LEE, H.T. and CILLIERS, J.J., (2002), "A foam drainage equation generalised for all liquid contents", *Journal of Physics: Condensed Matter*, **14**, 331-342

NEETHLING, S.J., LEE, H.T. and CILLIERS, J.J., (2003), "The recovery of liquid from flowing foams", *Journal of Physics: Condensed Matter*. **15,** 10, 1563-1576.

NEETHLING, S.J., LEE H.T. and GRASSIA, P., (2005), "The growth, drainage and breakdown of foams", *Colloids and Surfaces A*, **263**, 184-196.

VERBIST, G., WEAIRE, D., and KRAYNIK, A.M., (1996), "The foam drainage equation", *J. Phys.: Condens. Matter*, **8**, 3715-3731

WEAIRE, D. and HUTZLER, S., (1999). "*Physics of foam"*. Clarendon Press, Oxford, UK.