CFD MODELLING OF COLLISIONS AND ATTACHMENTS IN A FLOTATION CELL

P.T.L. Koh and M.P. Schwarz (CSIRO Minerals, Australia)
E-mail: Peter.Koh@csiro.au

In recent years, researchers have started to use computational fluid dynamic (CFD) modelling of mechanically stirred flotation cells to study the complexity of three-phase (air-water-solids) flows within the cells. Flotation cells are conventionally designed using empirically derived relations. In CFD modelling, the flotation cell is discretized into individual finite volumes where local values of flow properties are calculated. The detailed understanding of flow gained using this approach allows modification to existing equipment and operation to improve flotation performance.

The flotation effect may be studied as three sub-processes involving collision, attachment and detachment. A turbulent collision model has been used to estimate the rate of bubble-particle encounters, employing the local turbulent velocity, and the size and number concentrations of bubbles and particles obtained from CFD modelling. The collision rates in different parts of the cell have been calculated and compared. These rates represent the initial contacts between particles and bubbles in the cell. The probability of collision (collision efficiency) due to the streamline effect of fine particles moving around a larger bubble is estimated in the present work. An attachment rate based on the collision rate and collision efficiency is also calculated.