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

# MODELLING FLUID FLOWS IN SPLIT-FEED FEEDWELLS

**Robert B. WHITE[1], Ilija D. ŠUTALO[2], and Tuan V. NGUYEN[1]**

[1] Parker Centre for Integrated Hydrometallurgy Solutions
(CSIRO Minerals) Clayton, Victoria 3169, AUSTRALIA

[2] CSIRO Manufacturing and Materials Technology, Highett, Victoria 3190, AUSTRALIA

## ABSTRACT

Single-phase turbulent fluid flow inside a split-feed feedwell model was investigated in a small-scale physical model of the feedwell and simplified thickener to better understand the fluid flow process occurring inside and below split-feed feedwells. Laser Doppler velocimetry was used to measure velocity fields inside a split-feed feedwell.

The flow field in the split-feed feedwell is complex, the two tangentially opposed feed streams impact each other on the opposite side of the feedwell from the entry side. The high pressure at this impact point forces the combined stream to exit vertically downward, there is also a re-entrainment stream that provides a source of dilution for the feed.

A three-dimensional computational fluid dynamics (CFD) model was used to simulate the fluid flow through split-feed feedwell models. The effect of changing the split ratio and inlet pipe angles was investigated. Reasonable agreement with the experimental velocity results measured inside the feedwell was obtained, indicating the CFD model could be used to assist in the design of split-feed feedwells for industrial thickeners.

## INTRODUCTION

In an earlier review of flow measurements in scale models of clarifiers or thickeners White et al., (2003) showed that the investigations generally focussed on fluid flow in the body rather than inside the feedwell, that the errors associated with the velocity measurement techniques were usually 20% or higher and that the feedwell designs were usually oversimplified. However, Šutalo et al., (2000, 2001) and White et al., (2003) carried out velocity measurement in more realistic feedwell geometries using laser based techniques that had lower errors at around 3%.

There have been several two- and three-dimensional mathematical models developed to simulate the flow through wastewater clarifiers, but their feedwell designs were usually oversimplified (Krebs et al., 1995, McCorquodale and Zhou (1993), Matko et al., 1996, Jayanti and Narayanan (2004), Szalai et al., 1994, Frost et al., 1993, Quarini et al., 1996, Deininger et al., 1998, Lakehal et al., 1999, Brouckaert and Buckley (1999)). Their feedwell designs also differed and were often simpler than those encountered in the minerals processing industry. In general, the predicted flow fields inside clarifiers were found to vary significantly depending on the inlet design, diurnal variations in inlet flow and inlet solids loading.

Krebs et al., (1996) investigated the effect of inlet and outlet configuration on the flow in secondary clarifiers. Brouckaert and Buckley (1999) showed that addition of baffles at the skirt reduced the recirculation near the outlet weir. Quarini et al., 1996 found large dead spaces within sedimentation tanks. Deininger et al., 1998 investigated velocity and solid distributions in circular secondary clarifiers. They found circulating currents in both the radial and vertical directions. Jayanti and Narayanan (2004) investigated three types of inlets (straight circular, tulip type and tulip type with conical deflector) and found that the upper and lower cut-off sizes for a sedimentation tank are reduced when using a tulip type of inlet with a conical deflector compared to straight deflector.

White et al., (2003) modelled a single tangential entry feedwell with and without a shelf. The numerical predictions of the internal velocity fields were in good agreement with the measured results and the feed was found to spiral down the inner wall of the feedwell before exiting. The presence of an internal shelf resulted in an increase in the fluid residence time in the feedwell.

The split-feed feedwell was designed to increase the dissipation of the kinetic energy of the inlet feed stream, increase the residence time, to encourage uniform settling with minimum turbulence and to reduce short-circuiting in the thickener.

Flow structures in the split-feed feedwell are well known by users to be very sensitive to inlet conditions and are believed to be a result of the interaction between the two inlet streams.

The objectives of this study are as follows:

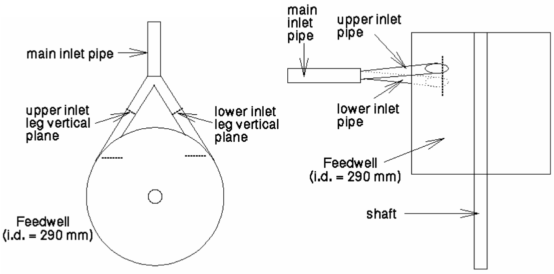
- To better understand the flow patterns in split-feed feedwells.
- To discuss the effect of varying the split ratio and inlet pipe angles.

## EXPERIMENTAL RIG AND TECHNIQUE

### Physical model

An optically transparent model of a flat-bottomed thickener with a central split-feed feedwell was constructed to carry out laser Doppler velocimetry (LDV) velocity measurements in the split-feed feedwell. The entire thickener model was placed inside a rectangular water bath in order to reduce optical distortion. An ONGA 112 pump was used to deliver water to the feedwell at flow rates up to 120 l min$^{-1}$. The feedwell has a diameter and height of 0.3 m, while the thickener had a diameter of 1.2 m. The central shaft has a diameter of

0.032 m. Both tangential inlets to the feedwell model are fed from a common inlet feed pipe with an internal diameter of 0.032 m. A schematic of the split-feed feedwell model is shown in Figure 1. Both inlets are fed from a Y-piece, and both branches of the Y-piece enter the feedwell at different elevations. The centre-line of the lower and upper inlets are off-set by approximately one inlet-pipe diameter in the vertical direction. A transparent lid was placed over the thickener model to reduce optical distortion and to support the feedwell and central shaft. The flow rate was varied using a variable speed controller and was measured using a magnetic flow meter.




(a) top view (b) side view

**Figure 1:** Schematic of the split-feed feedwell.

## Experimental technique

A detailed description of the LDV velocity measurement techniques used in this investigation is given in White et al., (2003). The time averaged velocity field and turbulence field were measured in a small-scale feedwell model on two vertical planes (perpendicular to each other). The time-averaged velocity results were shown to be reproducible (typically within 3%). In each leg of the split-feed feedwell inlet pipes 29 velocity measurements were made on a grid of points covering the pipe cross-section and averaged. The small pipe diameters of the inlet pipes (i.e. high curvature) made it difficult to measure velocity near the inside walls of the pipe. In the experiments the flow rate was 62 l min$^{-1}$, which corresponds to an average velocity of 1.285 ms$^{-1}$ and a feed pipe Reynolds number of 38,000.

### Numerical Modelling

The commercial software package CFX4 was used to solve the governing Navier-Stokes equations. The single-phase CFD model was developed for the whole thickener, including the cylindrical thickener body, feedwell and central shaft. The mesh contains 211,200 cells and the CFD model uses a similar fine computational mesh as was used in White et al., (2003), although the location of some surface patches has been altered to represent the split-feed geometry. The k-epsilon turbulence model was used to simulate turbulence.

## RESULTS

### Experimental results

LDV velocity measurements were undertaken to determine the exact split in the volumetric flow rate between the lower and upper inlet legs. The flow split was determined to be 38.6% (lower inlet leg): 61.4% (upper inlet leg). The velocities in each inlet pipe leg were higher on the side closer to the vertical plane passing through the main inlet pipe and feedwell centre. This is caused by the split in which the high speed fluid in the centre of the main feed pipe preferentially travels down the walls of the twin legs closest to the centre.

High average turbulence intensities were found in both inlet pipe legs (67% turbulence intensity in the lower pipe and 40% in the upper pipe) indicating the flow was not fully developed, most likely a consequence of the bifurcation in the feedpipe. Flow instabilities are therefore more likely in the split-feed feedwell than in an open single entry tangential feedwell because of these velocity fluctuations in the inlet.

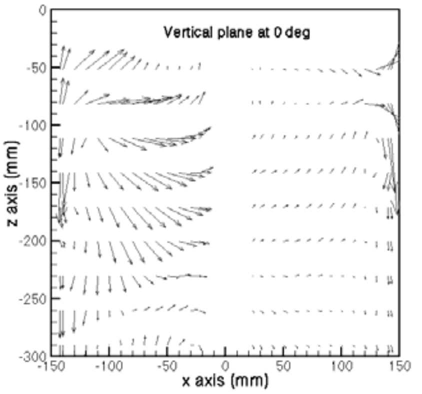
LDV velocity measurements were also carried out to determine the flow fields in two vertical planes through the split-feed feedwell and these were used as validation data for the CFD model. The measured velocity fields inside the split-feed feedwell shown in Figure 2a and 2b indicate that the flow is very complex.

The high turbulence intensities imply that the variation between subsequent LDV measurements made at a particular location is considerably greater than the variation between subsequent measurements made in a more stable flow field such as that found for a tangentially single entry feedwell. Considerably larger data sets than usual were therefore needed to produce average measurements with good statistics because a wide variation in measurements in a data set with too few sample points can results in an average measurement with an unacceptably high standard deviation.

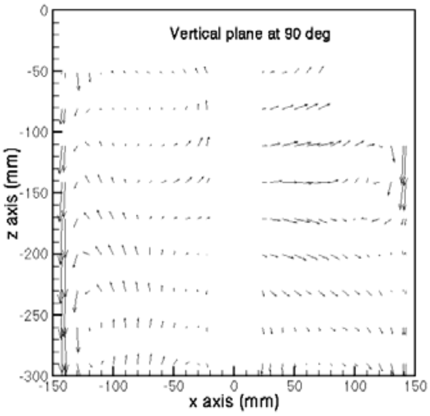
This experimental work has verified that there is considerable interaction between the two inlet streams, which confirms observations that the flow structures in the split-feed feedwell are very sensitive to inlet conditions. The two tangentially opposed feed streams have been observed to impact each other on the opposite side of the feedwell from the entry side and the high pressure at this impact point forces the combined stream to exit vertically downward on this side of the feedwell. This is clearly evident in Figure 2b where the large vertically downward flow can be seen on the left side of the feedwell opposite the entry side. There is also a re-entrainment stream that provides a source of dilution for the feed in the upper region of the feedwell, but it appears that the majority of the feedwell volume is not fully utilised.

### Model validation

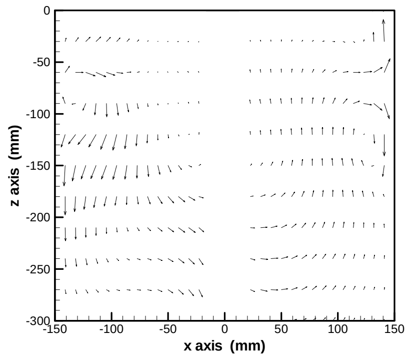
The experimentally determined split ratio and measured angle between each inlet and a normal to the wall of the split-feed feedwell was used when setting up the inlet boundary conditions to validate the CFD model. The CFD model was able to resolve the main features of the flow inside this split-feed feedwell including the subtle flow reversals in the lower regions of both the 0° and 90° planes shown in Figure 2. The velocity vectors in the left hand central region of the 0° plane (perpendicular to the main inlet pipe) do not match as well as other regions, but comparisons in the 90° plane (parallel to the main inlet pipe) generally show a very good match between the LDV measurements and the CFD predictions.



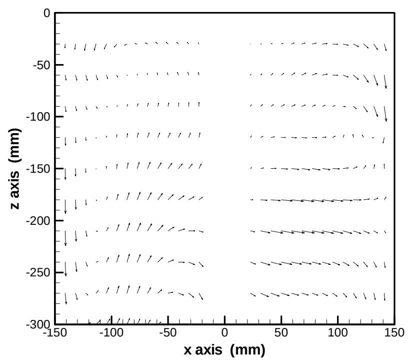

**a** – LDV experimental (perpendicular to inlet pipe)




**b** – LDV experimental (parallel to inlet pipe)




**c** – Predicted (perpendicular to inlet pipe)

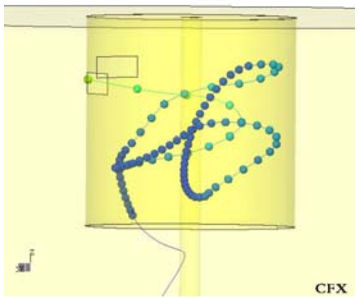



**d** – Predicted (parallel to inlet pipe)

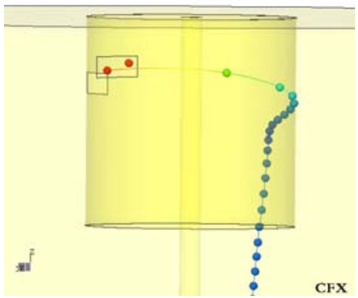
**Figure 2:** Experimental and predicted velocity fields in a split-feed feedwell model with flow split between the low inlet and high inlet of 38.6% to 61.4%.

The CFD model was found to be very sensitive to small changes in the geometry conditions at the inlet, so careful attention was placed on accurately modelling the inlet conditions to adequately simulate the flow conditions in the split-feed feedwell.

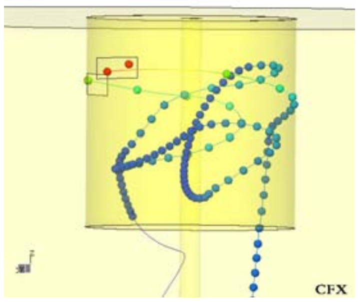
A numerical visualisation technique has been developed to provide more insight into the complex flow field in the split-feed feedwell. The technique involves using "streamlines" to identify the pathway taken by individual fluid volumes as they enter the feedwell through both inlets. A symbol was drawn at regular time intervals on the streamline to indicate velocity. The resulting image resembles a string of pearls that have one end fixed at a particular point on the inlet and the other end is free to move with the fluid flow as shown in Figure 3. This visualisation technique provides a better indication of the complex flow conditions in the split-feed feedwell, and also provides an indication of the fluid residence time in the feedwell.




**a** - Low inlet (flow into page)



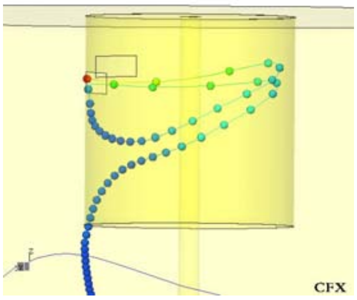

**b** - High inlet (flow out of page)



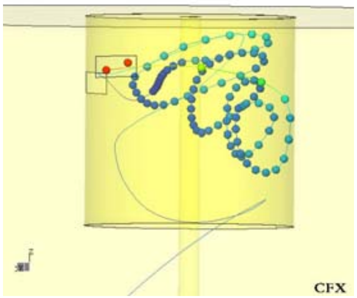

**c** - Both inlets

**Figure 3**: Streamlines showing flow direction and velocity in the split-feed feedwell run with a flow split between the low inlet and high inlet of 38.6% to 61.4% and an angle of deviation for the low inlet of 3.44° and an angle of deviation for the high inlet of -3.44°.

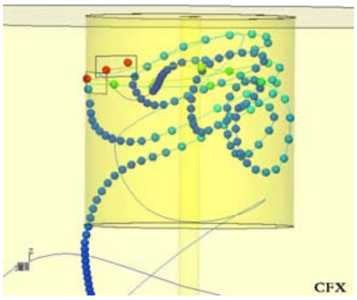
The numerical visualisation technique clearly shows similarity between the CFD results and the measurements made in the physical experiments. The large vertically downward flow evident in Figure 2b are also in Figure 3b, and the flow path identified in Figure 3a can account for the re-entrainment stream that was identified earlier.




**a -** Low inlet (flow into page)




**b -** High inlet (flow out of page)




**c –** Both inlets

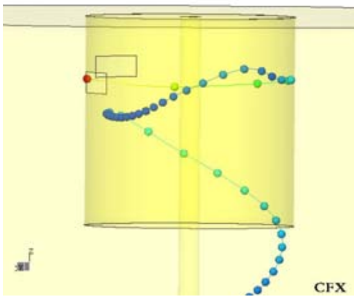
**Figure 4**: Streamlines showing flow direction and velocity in the split-feed feedwell at a flow split between the low inlet and high inlet of 50% to 50%.

#### Effect of flow split ratio

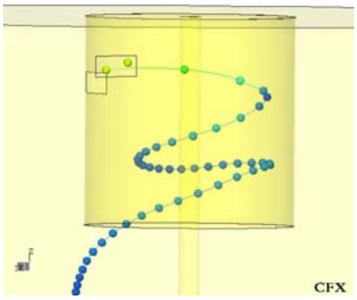
It was noted earlier that the internal flow patterns are very sensitive to the inlet conditions, and this is particularly easy to investigate using a CFD model. The results from two additional CFD runs with different flow splits are shown using the numerical visualisation technique presented here. The results for a flow split between the low inlet and high inlet of 50% to 50% are shown in Figure 4 and the results for a flow split between the low inlet and high inlet of 60% to 40% are shown in Figure 5.

Comparison of the predicted streamlines from each inlet at each of the three flow splits provides some very interesting insights into the internal flow patterns in the feedwell.

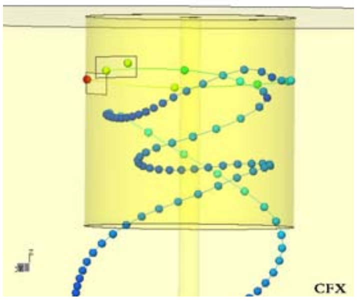
- By comparing Figures 3a, 4a and 5a, it appears that the residence time of the fluid entering through the lower inlet slightly increases as its proportion to the total flow is reduced.
- By comparing Figures 3b, 4b and 5b, it appears that the residence time of the fluid entering through the upper inlet more strongly increases as its proportion to the total flow is reduced.
- By comparing Figures 3c, 4c and 5c, it appears that a flow split between the low inlet and high inlet of 50% to 50% maximises the residence time of fluid in the feedwell, but a flow split of 38.6%:61.4% (lower:upper) split, provides a significantly longer residence time than a flow split of 60%:40%.




**a -** Low inlet (flow into page)




**b -** High inlet (flow out of page)




**c -** Both inlets

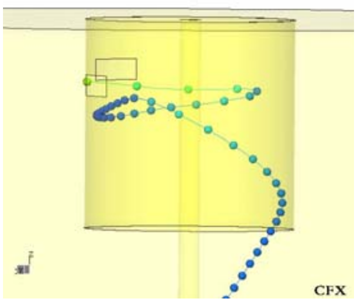
**Figure 5**: Streamlines showing flow direction and velocity in the split-feed feedwell at a flow split between low inlet and high inlet of 60% to 40%.

The residence time in the feedwell and the mixing pattern resulting from this interaction changed with split ratio, however the flow patterns through each inlet were quite different.

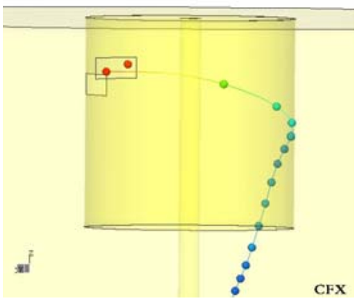
### Effect of upper and lower feedpipe angles

As noted earlier, both inlets are fed from a Y-piece, and since both branches of the Y-piece enter the feedwell at different elevations so that the centre-line of the lower and upper inlets are off-set by approximately one inlet-pipe diameter in the vertical direction. The angle of deviation from horizontal has been measured as +3.44° for the lower inlet and -3.44° for the upper inlet. These inlet angles have been used in the CFD runs discussed so far and, essentially, a positive angle of deviation from horizontal results in flow being directed downward and a negative angle results in flow being directed upward.

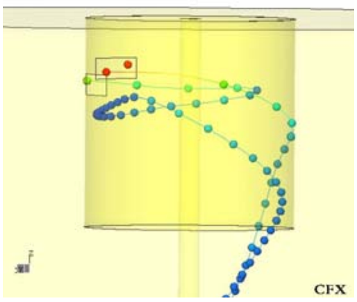
Since the internal flow patterns and residence times in the feedwell have been found to be very sensitive to the flow split ratio between each inlet, the CFD model has been used to investigate the sensitivity of the internal flow patterns to the angle of entry of each inlet.




**a** - Low inlet (flow into page)




**b** - High inlet (flow out of page)



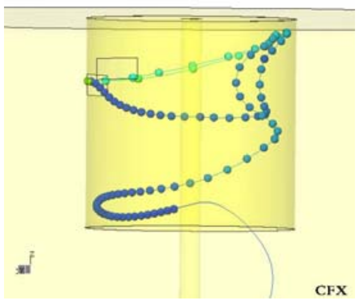

**c** - Both inlets

**Figure 6**: Streamlines showing flow direction and velocity in the split-feed feedwell at an angle of deviation for the low inlet of 0° and an angle of deviation for the high inlet of 0° at a flow split between the low inlet and high inlet of 38.6% to 61.4%.

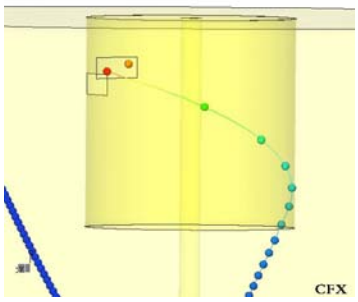
The results from two additional CFD runs with different angles of deviation for each inlet are shown using the numerical visualisation technique and can be compared with the validation run. The results for an angle of deviation for the low inlet of 0° and an angle of deviation for the high inlet of 0° are shown in Figure 6, while the results an angle of deviation for the low inlet of -15° and an angle of deviation for the high inlet of +15° are shown in Figure 7.

Comparison of the predicted streamlines from each inlet at each of the three inlet angles provides some very interesting insight into the internal flow patterns in the feedwell.

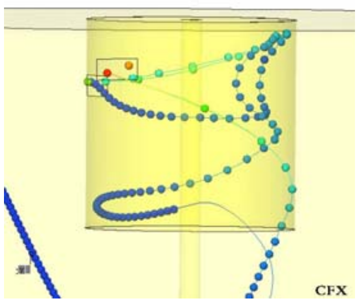
- By comparing Figures 3a, 6a and 7a, it appears that residence time of fluid entering through the lower inlet is significantly reduced by being directed in horizontally and there is only slight advantage in directing the flow upwards rather than downwards.
- By comparing Figures 3b, 6b and 7b, it appears that residence time of fluid entering through the upper inlet is not greatly influenced by the inlet angle.
- By comparing Figures 3c, 6c and 7c, it appears that a very low residence time of fluid in the feedwell is achieved when flows enter horizontally.




**a** - Low inlet (flow into page)




**b** - High inlet (flow out of page)




**c** - Both inlets

**Figure 7**: Streamlines showing flow direction and velocity in the split-feed feedwell at an angle of deviation for the

low inlet of -15° and an angle of deviation for the high inlet of +15° at a flow split between the low inlet and high inlet of 38.6% to 61.4%.

At the particular volumetric flow rates modelled (i.e., those of the validation run) it is difficult to draw many conclusions about the effect of non-parallel inlets. It is clear that there is considerable interaction between the flows entering the lower and upper inlets and it appears that as the residence time of one stream is increased then the residence time in the other stream is decreased because of the interaction between the two inlet streams.

## DISCUSSION

This CFD model has only been validated for single-phase flow against the split-feedwell model experimental results. It is expected that the presence of solids would alter the internal flow patterns, and so this work cannot be applied directly but rather can provide some indications as to the repercussions of operational and design decisions on overall thickener performance. The validated CFD model has been used to provide an indication of the possible changes in internal flow patterns that may result if the inlet angle was altered in the model feedwell. This provides some assistance in understanding the variations in flow that could result in a real thickener, where there is two-phase flow.

Alternative designs for the feedwell could include dedicated feed pipes for each inlet, and these inlets may be attached at a range of angles. It is also possible that flexible hoses could convey the feed to the feedwell, and the relative level of the hose may also change during normal operation and alter the effective angle at the feedwell inlet.

## CONCLUSIONS

The main conclusions from this investigation are:

- Complex flow interactions occur in the split-feed feedwell model.
- Acceptable agreement between LDV velocity measurements and CFD predictions of flow field can be achieved if appropriate attention to boundary and geometry conditions are included in the CFD model.
- Flow interactions in the split-feed feedwell model are very sensitive to the inlet conditions, in particular to split ratio and inlet angle.
- CFD models can be used to investigate the residence time of fluid streams entering the split-feed feedwell.
- Depending on the current operating conditions and design of a particular split-feed feedwell, the residence time of each stream entering the feedwell can vary significantly.
- The use of flexible hoses to introduce flow may result in horizontal flow into the split-feed feedwell, resulting in an unacceptable residence time in the feedwell.
- CFD modelling of a particular split-feed feedwell may greatly assist in the design and optimisation of thickeners.


## ACKNOWLEDGMENTS

This work was conducted as part of the AMIRA P266 'Improving Thickener Technology' series of projects. The authors wish to thank the following companies for their support: Albian Sands Energy, Alcoa World Alumina, Anglo Gold, Anglo Platinum, BHP Billiton, Cable Sands, Ciba Specialty Chemicals, Cytec Australia Holdings, De Beers Consolidated Mines, EIMCO Process Equipment, GL&V/Dorr Oliver, Glencore AG, Iluka Resources, Kumba Resources, Metso Minerals, Mt Isa Mines, Nabalco, ONDEO Nalco, Pasminco, Pechiney Aluminium, Queensland Alumina, Queensland Nickel, Rio Tinto, Tiwest, True North Energy, WMC Resources, and Worsley Alumina.


## REFERENCES


BROUCKAERT, C.J., BUCKLEY, C.A., (1999), "The use of computational fluid dynamics for improving the design and operation of water and wastewater treatment plants", *Water Science and Technology,* **40 (4-5)**, 81-89.

DEININGER, A., GÜNTHERT, F.W., WILDERER, P.A., (1996), "The influence of currents on circular secondary clarifier performance and design", *Water Science and Technology*, **34 (3–4)**, 405–412.

FROST, R.C., HALLIDAY. J., DEE, A.S., (1993), "Continuous Consolidation of Sludge in Large Scale Gravity Thickeners", *Water Science and Technology,* **28(1)**, 77–86.

KREBS, P., VISCHER, D., GUJER, W., (1995), "Inlet-structure design for final clarifiers", *Journal of Environmental Engineering,* **121 (8)**, 558–564.

LAKEHAL, D., KREBS, P., KRIJGSMAN, J., RODI, W., (1999), "Computing Shear Flow and Sludge Blanket in Secondary Clarifiers", *Journal of Hydraulic Engineering*, **135(3)**, 253-262.

MATKO, T., FAWCETT, N., SHARP, A., STEPHENSON, T., (1996), "A numerical model of flow in circular sedimentation tanks", *Process Safety and Environmental Protection,* **74 (B3)**, 197-204.

McCORQUODALE, J.A., ZHOU, S., (1993), "Effects of hydraulic and solids loading on clarifier performance", *Journal of Hydraulic Research*, **31(4)**, 461–478.

QUARINI, G., INNES, H., SMITH, M., WISE, D., (1996), "Hydrodynamic modelling of sedimentation tanks", *Proceedings of the Institution of Mechanical Engineers Part E-Journal of Process Mechanical Engineering* **210 (E2)**, 83-91.

ŠUTALO, I.D., NGUYEN, T., RUDMAN, M., (2000), "Modelling studies of fluid flow in a thickener feedwell model", *CHEMECA 2000*, Perth, Australia, 9–12 July.

ŠUTALO, I.D., NGUYEN, T., WHITE, R.B., RUDMAN, M., (2001), "Experimental and numerical investigations of the fluid flow in thickener feedwells", *Sixth World Congress of Chemical Engineering*, Melbourne, Australia, 23–27 September 2001, paper 941.

SZALAI, L., KREBS, P., RODI, W., (1994), "Simulation of flow in circular clarifiers with and without swirl", *Journal of Hydraulic Engineering*, **120 (1)**, 4-21.

WHITE, R.B., ŠUTALO, I.D., NGUYEN, T., (2003), "Fluid flow in thickener feedwell models", *Minerals Engineering* **16 (2)**, 145–150.