*Ninth International Conference on CFD in the Minerals and Process Industries*
*CSIRO, Melbourne, Australia*
*10-12 December 2012*

# ASSESSMENT OF THE FLOW PATTERN IN A SOLVENT EXTRACTION SETTLER

**Graeme LANE[1*], Krishna MOHANARANGAM[2] and William YANG[2]**

[1] CSIRO Mathematics, Informatics and Statistics, Clayton, Victoria 3169, AUSTRALIA
[2] CSIRO Process Science and Engineering, Clayton, Victoria 3169, AUSTRALIA
*Corresponding author, E-mail address: Graeme.Lane@csiro.au

## ABSTRACT

Mixer-settler units are used in solvent extraction processes in a number of industries, including hydrometallurgical plants. Efficient operation depends on the generation of a fine dispersion in the mixer section for metal transfer between immiscible aqueous and organic phases, followed by complete phase separation in the settler. However, settler performance depends critically on the flow pattern. The fluid flow in a settler at a nickel/cobalt plant was investigated through on-site measurements and computational fluid dynamics. Reasonable agreement was found between measurements and simulation results. It was found that the flow pattern of the current design is sub-optimal, exhibiting substantial short-circuiting and regions of recirculation, which may impact negatively on phase separation. Additional CFD simulations have been carried out to investigate modifications to the settler internals with the aim of improving the flow pattern. Calculations of the residence time distribution have shown that a revised design reduces early exiting of fluid.

## NOMENCLATURE

$c_D$ drag coefficient
$d_p$ droplet or particle diameter
$U_s$ slip velocity
Re Reynolds number

$\alpha$ volume fraction
$\rho$ density
$\mu$ dynamic viscosity

*Subscripts:*
$c$ continuous phase
$d$ dispersed phase

## INTRODUCTION

Solvent extraction is an important process in a number of industries, including hydrometallurgical plants, e.g. for the production of metals such as copper, nickel, cobalt and uranium. The process involves the contacting of an aqueous solution with an organic phase to achieve selective extraction and concentration of the metals of interest. In a large number of plants, the solvent extraction process is carried out using mixer-settler units, where a dispersion of one phase in the other is generated in one or more mixing tanks, followed by gravity separation in a settler. In the settler, efficient separation with minimal entrainment is critical to the economics of the process, and this depends on, amongst other things, the fluid flow pattern. The most efficient flow pattern to ensure sufficient time for separation of all droplets would be plug flow, but this is difficult to achieve in practice, and flow patterns often deviate significantly from this ideal, leading to reduced efficiency or throughput.

The importance of the flow pattern in settlers has been recognised for many years, and various efforts have been made to improve the flow pattern through changes to the design of settlers (Miller, 2006). A particular difficulty is that the inlet, coming from a mixer box, is typically much smaller than the width of the settler box, so it is difficult to make the incoming flow spread out across the box without introducing undesirable features such as turbulence, short-circuiting and flow reversals (Miller, 2006). The shallow depth of the inlet launder compared to the settler depth is also a cause of poor flow distribution. Various internals are used to attempt to improve the flow pattern, with the use of picket fences being very common.

CFD modelling provides a means of investigating the internal flow patterns in detail and assessing the effect of design changes. Miller (2006) reviewed several CFD studies which were able to shed light on internal flow patterns. For example, the CFD modelling of Kankaanpaa (2005) showed that in the absence of picket fences, the jet formed by the inlet penetrated for long distances, and also, the shallow depth of the inlet resulted in a reverse flow of the aqueous phase from half way down the settler back to the inlet end. The CFD modelling approach has also been used at Hatch (Poulter et al., 2011) to develop improved designs, such as a curved launder connecting the mixer and settler which is said to eliminate turbulence at the settler inlet. Hatch have also applied CFD modelling to develop ‘feed distribution arrays’ as an alternative to picket fences, and a bull-nose profile for the organic overflow weir which reduces entrainment of the aqueous phase.

This paper describes a study which was carried out to assess the flow pattern in an existing solvent extraction settler at a plant producing nickel and cobalt. As well as CFD modelling, on-site measurements of actual flow patterns have been carried out using an Ultrasonic Velocity Profile (UVP) probe. The investigation has identified a flow pattern which deviates significantly from ideal plug flow, with the implication being relatively poor performance for this settler. Consequently, design changes for the settler internals were investigated through additional CFD modelling with the aim of improving settler performance.

## DESCRIPTION OF THE SETTLER

The layout of the settler is illustrated in Figure 1. The settler has a width of 3.4 m and a length of 5.67 m from the inlet to the organic overflow weir. Aqueous/organic dispersion enters the settler from the mixer stage through a centrally positioned launder. The settler floor has a transverse slope of 1:100. The settler is fitted with two picket fences. In each picket fence there are two rows of pickets, with seven pickets on the upstream side and thirteen pickets on the downstream side.

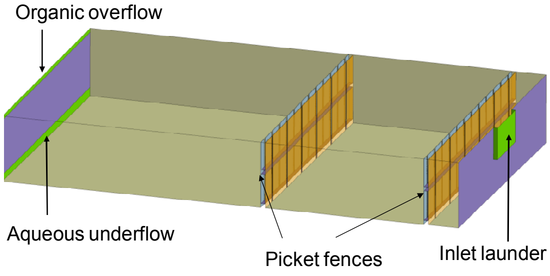


**Figure 1** Geometry of the settler (for base case) as generated for the CFD model.

## EXPERIMENTAL METHOD

A campaign of on-site measurements was undertaken at the plant, in which a Met-Flow UVP system was used to determine the flow field inside the settler. This system is capable of measuring instantaneous flow profiles along the axis of an ultrasonic beam by detecting the Doppler shift frequency of ultrasound echoes as a function of time. The UVP system has the benefit that it can be used with opaque fluids such as the actual organic and aqueous phases in an industrial solvent extraction plant, whereas other commonly used flow measurement techniques such as Laser Doppler Velocimetry (LDV) and Particle Image Velocimetry (PIV) are not applicable in this situation. Furthermore, since the Doppler shift frequency is directly related to the velocity magnitude, the system does not require calibration. The UVP transducers are relatively small, with a diameter of about 24 mm, so they can be placed inside an industrial settler without significant disturbance to the flow field which is to be measured.

A custom-designed holder frame was designed so as to be able to position several UVP transducers accurately within the settler simultaneously at two levels (in organic and aqueous layers). The UVP measurement system was used to determine the velocity field at 150 mm below the surface (in the organic layer) and 150 mm above the bottom (in the aqueous layer), for a grid of measurement points set approximately 300 mm apart. The UVP probes were also used to identify the position of the interface between the dispersion band and the organic layer.

## CFD METHOD

The CFD modelling method involves solving the equations governing fluid flow on a finite volume mesh which is generated for the geometry of interest. The finite volume mesh was a non-uniform hybrid mesh which was generated so as to provide higher grid resolution near the pickets where the flow pattern is more complex. The total mesh size was about 2.3 million elements. The two-phase Eulerian-Eulerian equations were solved using the ANSYS-CFX12 software. The *k*-epsilon model was included to model turbulence.

The liquid surface was assumed to be flat and a free-slip boundary condition was applied for the surface. Boundary walls were specified as zero slip. Uniform velocity and volume fraction were assumed for each phase at the inlet. Uniform flow was assumed for the aqueous and organic phase outlets.

Based on data supplied by the plant, the inlet flow rate for organic phase was assumed to be 65 $m^3$/hr. The flow rate of aqueous phase to the mixer/settler unit is 4 $m^3$/hr, but there is an internal recycle stream which increases the effective aqueous flow rate to the settler. This recycle stream was estimated to be about 40 $m^3$/hr and hence, the organic/aqueous feed ratio is about 60:40. The organic phase was assumed to be the continuous phase in the inlet stream.

Although some work has been undertaken to develop a modelling method to predict droplet size in settlers based on a number density equation, a more simplified approach was taken here due to the lack of any data to calibrate the coalescence rate in such a model. Instead, allowing for the expectation that the droplet size will be smallest at the inlet and then grow due to coalescence in the dispersion band, the droplet size was assumed to be 250 micron at the inlet and upstream of the first picket fence, and 450 micron downstream of the first picket fence.

The settling velocity within the dispersion band depends on the drag coefficient, $C_D$, and this was specified following the equation of Kumar and Hartland (1985) for liquid-liquid dispersions, which takes into account the volume fraction of the dispersed phase:

$$C_D = 0.53 + \frac{24}{\mathrm{Re}_p}\left(1 + 4.56\alpha_d^{0.73}\right) \tag{1}$$

where the Reynolds number is defined as:

$$\mathrm{Re}_p = \frac{\left(1-\alpha_d\right)\rho_c U_s d_p}{\mu_c} \tag{2}$$

where $\alpha_d$ is the dispersed phase volume fraction, $U_s$ is the slip velocity, $d_p$ is the particle diameter, $\mu_c$ is the continuous phase viscosity, and $\rho_c$ is the continuous phase density.

Additional simulations were carried out to investigate the effect of changes in the settler internals with the aim of improving the flow pattern. These simulations were based on modelling single-phase flow. This was considered to be justified because the observed single-phase flow pattern in the base case was quite similar to the two-phase flow pattern. Since the computational demand is reduced, this increased the number of alternative designs which could be tested within the available time.

As a means of comparing different flow patterns, additional runs were carried out to simulate a tracer, which permits calculation of residence time distributions. After obtaining the solution for the flow field for each configuration, the flow field was frozen and a time-stepping simulation was carried out for an additional scalar equation, in which the change in tracer concentration at the outlet was determined in response to a step change of tracer concentration at the inlet. The simulation was initiated in each case with a tracer concentration set to 100 (in arbitrary units) at the inlet and zero elsewhere. The concentration of the tracer at the outlets was monitored as a function of time, and a single mixed outlet concentration was obtained by adding the two outlet flows. The residence time curve (or exit age distribution) is then obtained as the first derivative of the curve for concentration as a function of time (Levenspiel, 1972).

## RESULTS AND DISCUSSION

### Base case design

Results are presented here for the CFD simulation results and comparison is made with the experimental measurements from the plant using the UVP probes. Figure 2 shows the profile for the organic volume fraction as plotted on several vertical planes. It can be seen that fluid enters the settler through the inlet as a mixed dispersion and then separates into three layers: an organic layer at the top, a dispersion band, and separated aqueous phase at bottom. The dispersion band is predicted to extend over the whole length of the settler, though with decreasing holdup of aqueous phase towards the outlet end. The thickness of the organic layer was in fairly good agreement with plant measurements.

The predicted flow pattern is illustrated in Figure 3, which shows velocity vectors in a horizontal plane 100 mm below the surface (in the organic layer). It can be seen that the liquid flows outwards from the inlet to each side and a recirculation is set up in the space before the first fence. Then, in the space between the first and second fences, the flow pattern exhibits forward flow along each side wall, and reverse flow back near the centre. Downstream of the second fence, the flow is mostly in the forward direction, but with a significant bias towards higher flow near the centre. The flow pattern shows similar features in the aqueous layer, but with lower velocities since the aqueous band is thicker and the aqueous volumetric flow rate is lower. The flow pattern is asymmetrical in some places, e.g. there is more forward flow on one side after the first fence. This asymmetry is presumably due to the transverse slope of the settler bottom (where the settler is deeper at one side).

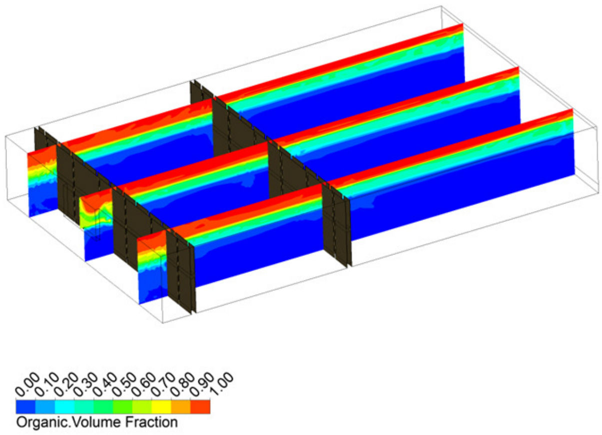


**Figure 2** Volume fraction of organic phase in selected horizontal planes (inlet is at left, outlet at right).

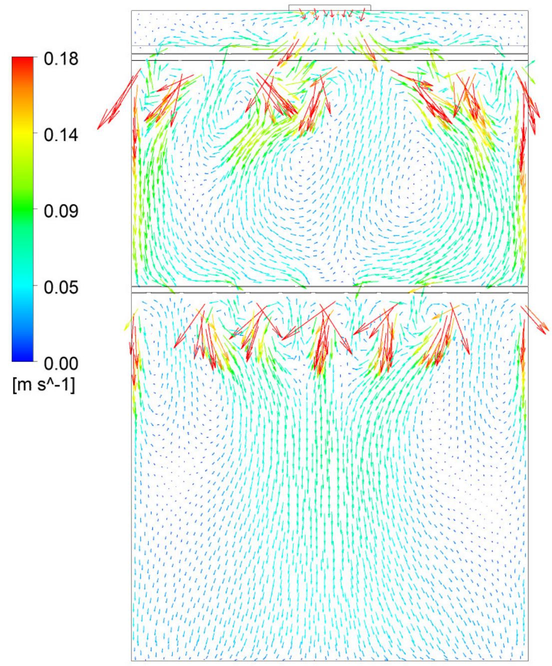


**Figure 3** CFD simulation of base case design: velocity vectors in organic layer near surface.

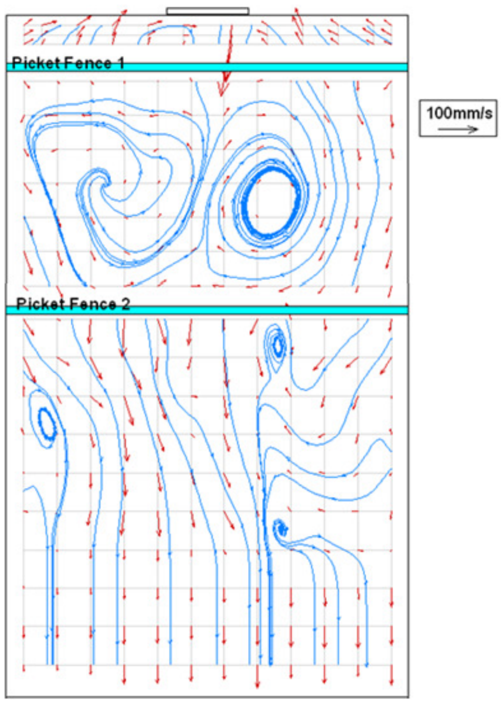


**Figure 4** Velocity field and streamlines at 100 mm below surface, as measured on plant using UVP probe.

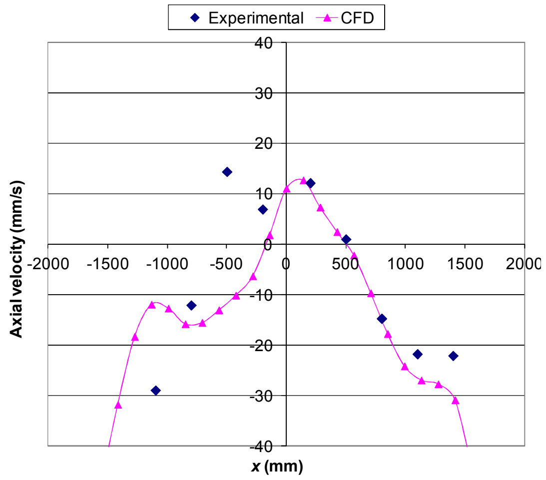


**Figure 5** Axial velocity component as a function of transverse position, *x*, at 2.055 m from inlet end (in organic layer).

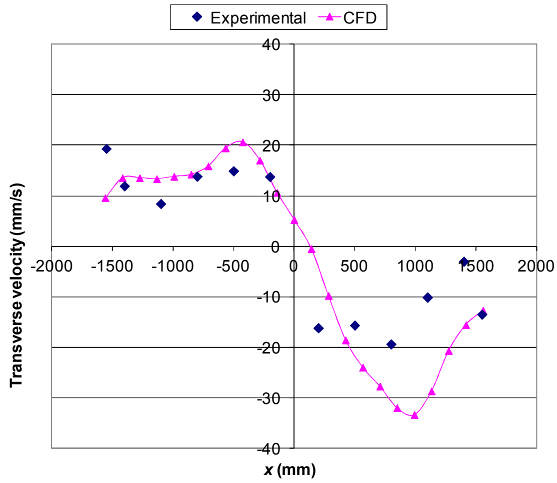


**Figure 6** Transverse velocity component as a function of transverse position, *x*, at 2.055 m from inlet end (in organic layer).

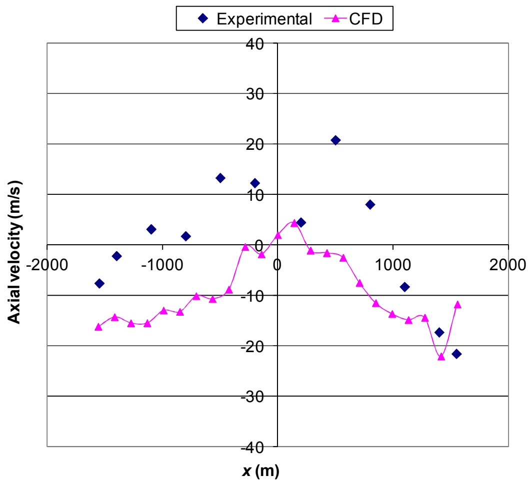


**Figure 7** Axial velocity component as a function of transverse position, *x*, at 2.055 m from inlet end (in aqueous layer).

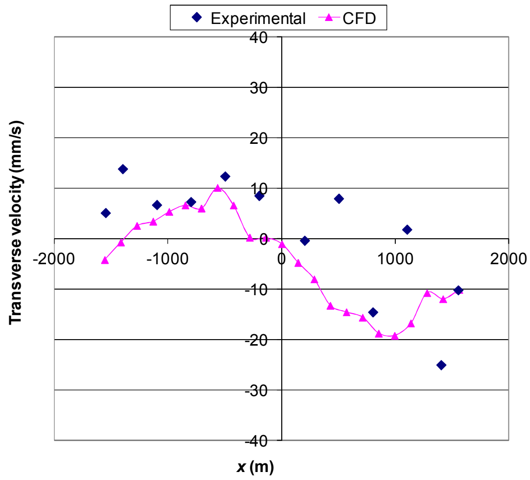


**Figure 8** Transverse velocity component as a function of transverse position, *x*, at 2.055 m from inlet end and at 200 mm above bottom (in aqueous layer).

The fluid flow pattern according to the CFD model can be compared with experimental measurements, such as in Figure 4, which shows the measured flow pattern in the organic layer. Qualitatively, all the main features as predicted by the CFD model can be observed in the measured flow field, including reverse flow towards the inlet near the inlet end wall, preferential forward flow along the wall between the two fences with reverse flow near the centre, and preferential forward flow towards the centre after the second fence. More quantitative comparisons have been carried out by plotting velocity profiles at selected locations. Profiles at 2.055 m from the inlet end in organic and aqueous layers are shown in Figures 5 – 8. Agreement is generally very good for the organic layer, but not quite as good in the aqueous layer.

The comparison between CFD and plant measurements was complicated by the fact that the plant conditions did not match up exactly with the assumptions of the CFD model. The CFD model was based on 'clean' geometry according to engineering drawings, whereas at the time of measurements a layer of sediment about 100 mm thick was detected on the settler bottom and the first picket fence was found to be out of alignment. Also, CFD modelling was carried out in advance of the experimental campaign and operating conditions were based on historical data which did not match up exactly with the actual plant conditions at time of measurement, e.g. the measured overall liquid depth was 0.1 m less than assumed in the CFD model and the organic flow rate was about 30% lower compared to the values assumed in the CFD model. Given the various discrepancies, the general agreement between CFD and measurements seems quite reasonable.

It can be concluded that the settler exhibits a flow pattern which is detrimental to performance in several ways. Reverse flow may tend to re-mix separated aqueous and organic phases back into the dispersion layer, while other regions where flow is in the forward direction but at velocities considerably higher than the average superficial velocity indicate short-circuiting, meaning that for a portion of the feed, there is a reduced time available for settling out of droplets. The residence time distribution for the base case has been calculated (see Figure 9), and this was found to be very wide. Early exiting is evident, with injected tracer in the CFD model first arriving at the outlet after about 130 seconds, compared to a mean residence time of about 610 seconds.

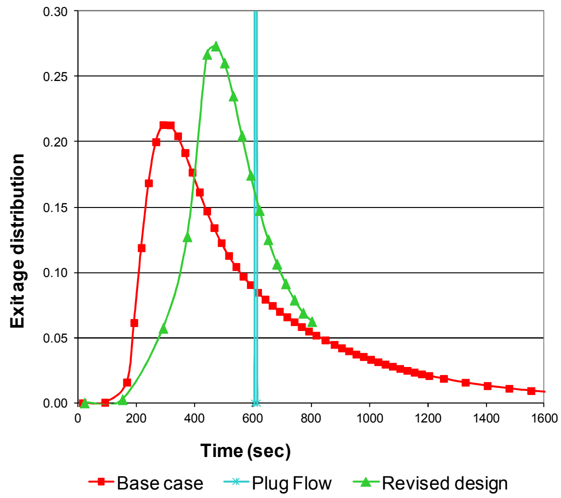


**Figure 9** Exit age distribution for base case and revised design (single phase simulations). Theoretical value for plug flow is also shown.

### Investigation of alternative designs

Further CFD modelling was carried out to investigate alternative configurations of the settler internals with the aim of improving the flow pattern. In the main part, this study involved testing of changes to the picket fences, such as the number of pickets, the gap width between pickets, the positions of the fences and the number of fences. Several practical constraints were kept in mind. One constraint wa-s that the gap between pickets should not be too small, since small gaps can become blocked due to build-up of crud or solids. Therefore, it was decided that the gaps in each fence should not be any smaller than the smallest gap in the base case. Another important constraint is that the overall pressure drop should not exceed that of the base case, since the plant would not be able to maintain the same flow rates without modification to the mixer stage, which provides the pumping head for the system.

One change which was thought to have potential benefit was to increase the number of pickets, since strong jets are formed on the discharge side of fences due to the limited number of gaps in the fences. These jets are thought to be a significant source of poor flow distribution, since it is observed that the individual jets join together into larger scale flow features, which set up flow reversals. However, increasing the number of gaps was not initially as beneficial as expected because this also leads to reduced pressure drop due to higher open area, and it is thought that maintaining pressure drop also assists in achieving flow uniformity. However, the reduction in pressure drop per fence opened up the opportunity to add a third fence without exceeding the overall pressure drop in the base case, and this was found to improve the flow pattern. Another modification was the addition of angled vanes at the inlet to spread the incoming flow more towards the sides.

The flow pattern for one alternative design is illustrated in Figure 10. In this configuration, the settler is fitted with three picket fences. The number of pickets in each fence has been increased, so that in each picket fence there are twelve pickets on the upstream side and twenty-three pickets in the downstream side. Angled distributing vanes were also added adjacent to the inlet. With this design, a more uniform flow field is obtained overall. This can be assessed in terms of the mean deviation of local velocities from the average superficial velocity. For the base case, the mean deviation was calculated to be 0.018 m/s, but for this revised design, the mean deviation reduces to 0.011 m/s. Also, because the fences have a higher fraction of open area compared to the base case, they generate a smaller pressure drop, being about 55% of the base case value. Thus, the additional fence does not compromise the pumping capacity of the system.

The residence time distribution for this revised configuration is shown in Figure 9. The curve shows a sharper peak with a maximum value closer to the plug flow value, and the proportion of fluid with early exiting times is reduced. Thus, this RTD curve suggests an improvement in flow pattern for this design, and this would be expected to lead to improved settling capacity and reduced droplet entrainment.

Further work is proposed to look at other possible designs for the settler internals, with the aim of achieving even greater improvement. In particular, since the overall pressure drop has been reduced substantially, there is the possibility of adding an additional picket fence, and this modification should be investigated further.

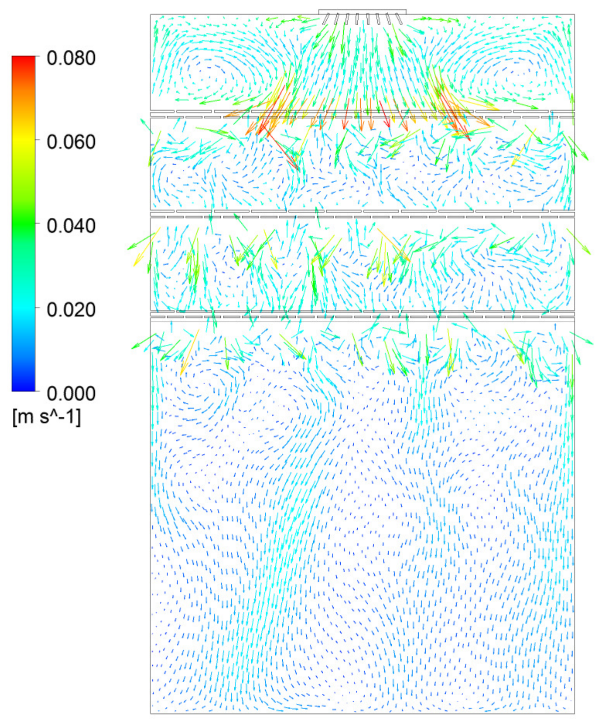


**Figure 10** Revised design with three picket fences: velocity vectors at 0.4 m above bottom

## CONCLUSION

The flow pattern in a solvent extraction settler has been assessed through CFD modelling and plant measurements using UVP probes. The two data sets match reasonably well. These studies have revealed a sub-optimal flow pattern in the current design. Further CFD modelling was undertaken to assess alternative designs for the internals of the settler. A revised design has been presented here which incorporates three picket fences, with an increased number of pickets and gaps in each row, and additional angled vanes at the inlet. CFD simulation indicates a more uniform flow pattern, and the residence time distribution indicates a reduction in early exiting of fluid. This design would be expected to lead to improved settling capacity and reduced droplet entrainment. It is also suggested that further improvements should be investigated. For example, since the overall pressure drop has been reduced, there is scope to add an additional picket fence without compromising the pumping capacity of the system.

## ACKNOWLEDGEMENTS

The authors wish to thank MERIWA and the commercial sponsors of the Solvent Extraction Technology projects (SXT and SXT2) for their support, and also acknowledge the support of the CSIRO Minerals Down Under National Research Flagship and Parker CRC for Integrated Hydrometallurgy Solutions.

## REFERENCES

KANKAANPAA, T., (2005), “Studying Solvent Extraction Settler Process by Using CFD”, EPD Congress, 2005 (Schlesinger, M.E., ed.), TMS.

KUMAR, A. and HARTLAND, S., (1985), “Gravity Settling in Liquid/Liquid Dispersions”, *Can. J. Chem.* **63**, 368.

LEVENSPIEL, O., (1972), *Chemical Reaction Engineering*, 2nd edition, Wiley International.

MILLER, G., (2006), “Design of mixer-settlers to maximise performance”, ALTA Copper 10, May 2006.

POULTER, S., ROBLES, E., PANAOU, C., and GOTTLIEBSEN, K., ( 2011), “Hatch customised solvent extraction technology”, *ISEC 2011*, 19th International Solvent Extraction Conference, Santiago, Chile, 3 – 7 October, 2011.

 