Second International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
6-8 December 1999

COMPARISON BETWEEN CFD AND MEASURED DATA FOR THE MIXING OF LEAD BULLION

C. BAILEY[1], S. KUMAR[1], M PATEL[1] A.W PIPER[2], R A FORSDICK[2], S HANCE[2]

[1]Centre for Numerical Modelling and Process Analysis, University of Greenwich, London, UK

[2]Britannia Refined Metals Ltd, Northfleet, UK

ABSTRACT

The refining of lead bullion takes place in hemispherical vessels (known as kettles) of varying sizes. It is normal practice to remove impurity elements (i.e. copper, silver, bismuth, antimony, etc) sequentially, by the addition of reagents, which selectively react with the impurity element. Dross, containing these elements, forms on the lead bath surface, which is then removed at the end of the reaction period for subsequent processing. Unfortunately, the dross content will contain a large amount of lead as well as the impurity element. Mixing this dross back into the kettle provides a mechanism to remove the lead and hopefully capture more of the impurity, still present in the lead bath. The efficiency of this mixing process is dependent on parameters such as impeller design, shaft depth and shaft speed as these will govern the dross grades.

Computational fluid dynamics (CFD) is playing a key role in helping understand and hence optimize this mixing process. This paper presents data comparing CFD predictions for velocity in a kettle with data gathered on a scaled water model and a real lead-kettle.

NOMENCLATURE

p pressure
u_i velocity component
ρ density
τ_{ij} stress tensor
ω rotational reference frame

INTRODUCTION

Batch mixing of materials, with impellers, is a process used extensively in a number of industrial sectors (Edwards, 1985). For lead refining, a number of factors govern the degree of efficiency of the kettle mixing process, including the dross yield, lead oxidation, power consumption, time taken to mix, and surface swirl. At Britannia Refined Metals mixing is achieved using either 3 or 4 pitch bladed impellers, which generate a vortex in the central region of the kettle. This vortex provides a mechanism that pulls buoyant dross down into the kettle where it can react to refine the bath. Throughout the industry a wide range of mixer types are used which are all trying to achieve the same result - removal of impurities from the lead with minimum lead content in the developed dross. Establishing mixing conditions that maximise dross grade will significantly help reduce costs.

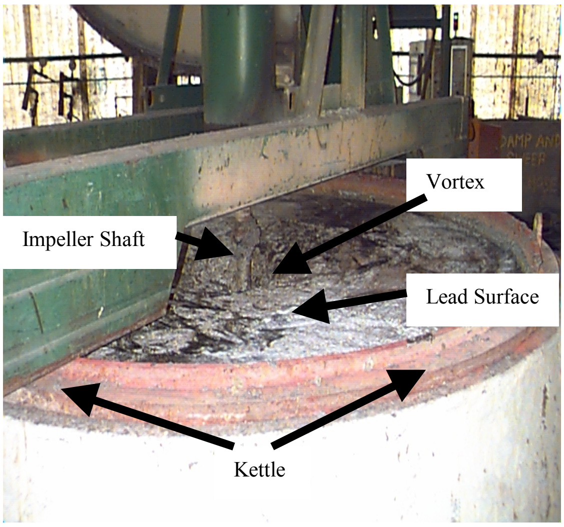

Figure 1: Lead refining operation using a kettle

Figure 1 details the surface of a kettle where a bridge mixer is being used to stir the lead bath. Figure 2 shows the shape of the kettle and the type of flow patterns that may occur during mixing. Although figure 2 shows flow patterns that indicate good mixing throughout the vessel, it is unclear whether this happens in practice.

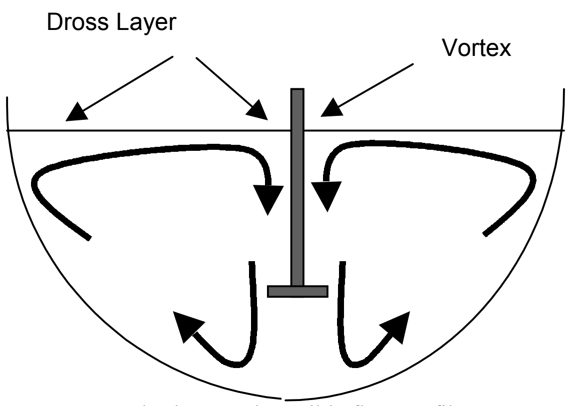

Figure 2: Kettle shape and possible flow profiles.

COMPUTATIONAL FLUID DYNAMICS

The main complication for using CFD to predict flow in an agitated vessel is representing the impeller blade geometry in detail, as well as the momentum from the impeller blades to the surrounding fluid during mixing. Up until the early 1990's the majority of mixing simulations treated the impeller region as a black box (Bakker and Van Den Akker, 1992), where its action was included into the simulation using experimental data. Only recently has full three-dimensional modelling occurred that predicts both the flow in the vessel and between the blades (CY Perng and JY Murthy, 1993; Rande and Dommeti, 1996). To achieve this, a rotational frame of reference is used around the impeller region where the Coriolis acceleration and Centrifugal force are included in the momentum equations. These approaches have also allowed simulations to be undertaken with fixed baffles in the vessel. In this paper we have adopted the Fluent CFD code (Fluent V4.5) to simulate the mixing of lead inside a kettle. Currently, we are solving the lead mixing process as a steady state single-phase solution that, as a starting point, will allow us to validate the models with gathered data.

Governing equations for mass and momentum conservation in rotating frame of reference are given in Cartesian tensor as

$$\frac{\partial \rho u_i}{\partial x_i} = 0.0$$

$$\frac{\partial \rho u_j u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} - 2\rho\varepsilon_{ijk}\omega_j u_k + \rho\omega_j(\omega_j x_i - \omega_i x_j)$$

Where x_i represents the i-th Cartesian coordinate, u_i the i-th Cartesian velocity component in a reference frame rotating with ω rotational speed, ρ the density of the fluid, P the pressure, and ε_{ijk} is the alternating unit tensor. In the above equation for momentum, the term $2\rho\varepsilon_{ijk}\omega_j u_i$ is the Coriolis acceleration and $\rho\omega_j$ (ω_j x_i - ω_i x_j) is centrifugal force acting due to the rotation of the reference frame. Here standard k-ε model has been used to account for the turbulence in the flow field and hence, stress tensor τ_{ij} in this case is given by

$$\tau_{ij} = (\mu + \mu_t)\left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right]$$

Where δ_{ij} is the Kronecker delta, μ and $\mu_t = C_\mu \rho k^2 / \varepsilon$ are molecular and turbulent viscosity respectively with k as turbulent kinetic energy , ε as dissipation rate of the turbulent kinetic energy and C_μ as modelling constant. These governing equations are integrated over a body-fitted structured mesh representing the kettle. Figure 3 details such a mesh used for a two-baffled kettle design.

At present very little CFD modelling has been undertaken on the mixing of lead using kettles. Kumar et-al (1999) presented details on gathered plant data and the effect of operating conditions on power consumption and vortex formation.

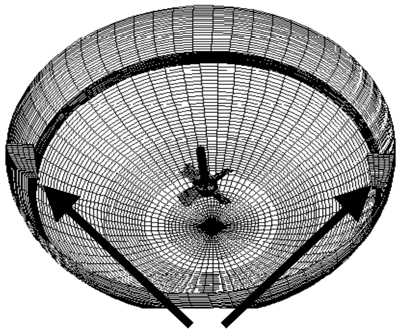

Figure 3: Kettle model with baffles.

WATER MODELLING

Figure 4 shows a 0.75 scaled water model for the kettle design under investigation.

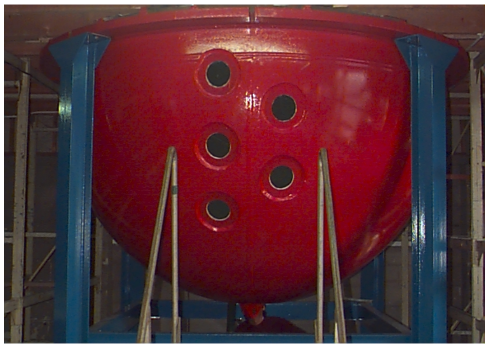

Figure 4: Water Model.

The above water model is providing visual data on the mixing performance of the kettle for a number of operating conditions. Using similitude, we can match Froude numbers for the flow of water and lead. This provides an opportunity to then observe the flow patterns in the water model, which, with the appropriate scaling, will approximate those found in a real lead-kettle Also, velocity data is being gathered at different points in the water model to help validate the CFD model. A probe (NORTEK), based on the acoustic doppler principle, is being used to obtain this data. Figure 5 shows the set-up used to obtain velocity data from the kettle.

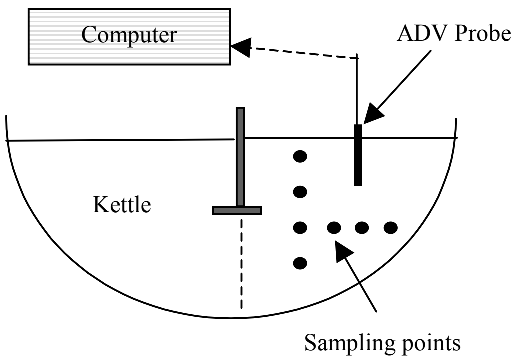

Figure 5: Schematic diagram of geometry.

Unlike a lead mixing process, baffles were also placed in the water model to investigate different flow regimes. Data was gathered at each sampling point after the flow had developed for ten minutes using a shaft speed of 140rpm. This data was then stored on computer at a sampling rate of 20Hz for a kettle design with and without baffles. A centered impeller shaft with four pitched blades of 45 degrees was used for all the tests.

GATHERING PLANT DATA

As well as gathering data from the water model, we have also gathered surface velocity data from a real 200tonne hemispherical kettle. A hole was cut into the mixer frame to create a window where a digital camera was placed to observe the movement of the lead surface. Figure 6 shows a frame from the video camera of the lead surface near the impeller shaft through the hole cut into the mixer. To obtain an estimate of the surface velocity, a steel ball bearing was dropped into the lead at the kettle rim. The progress of this marker was then timed and monitored visually as it moved towards the center of the vessel.

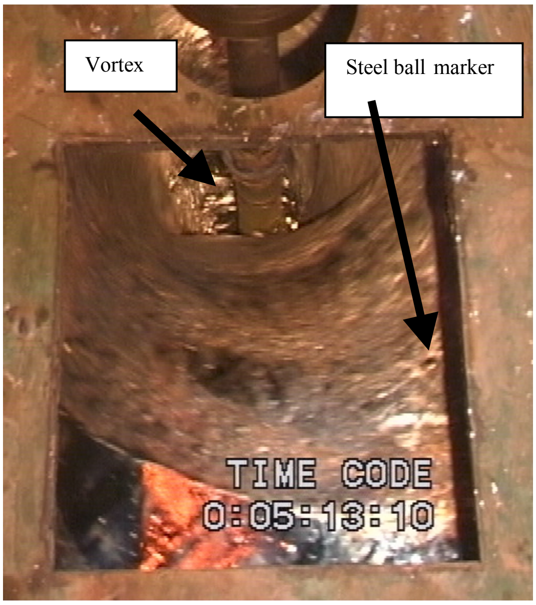

Figure 6: Video capture of marker on lead surface.

As it passed below the hole in the mixer its movement was captured using the camera. This footage, with the location and angle of the camera and the location and dimensions of the hole, provides an estimate of the steel ball velocity. Although the density of steel is lower than lead, this calculation can be used to estimate lead surface velocity at the location of the steel ball. This estimate can then be used for model validation.

RESULTS

As detailed above, velocity data has been gathered from both the water model and a real lead-kettle. The Fluent CFD code has been used to predict the flow profiles for the water model, where baffles have also been used, and also for the real kettle which did not have baffles.

No-slip boundary conditions are used at all solid walls, and a flat surface is also assumed at the top of the kettle. For both the water model and the real-kettle the shaft was placed at half kettle depth. The shaft velocity used in the water models was 140rpm and that used for the real kettle is 80rpm. Also, unlike the water model test, the real-kettle test used flat blades on the impeller. A high-resolution mesh density of 67,200 and 65,000 elements was used for the real-kettle and water model simulations respectively. Also, the two-equation k-ε turbulence model was used to predict turbulence throughout the domain.

Water Model Comparisons

A kettle design with and without baffles has been modelled using the Fluent CFD program. Figure 7 shows the velocity profile at the top surface for the water model without baffles. Clearly it can be seen that for this design the flow profile is circular, essentially travelling around the shaft with little movement towards the center. This type of flow profile is also observed visually during the water model trials.

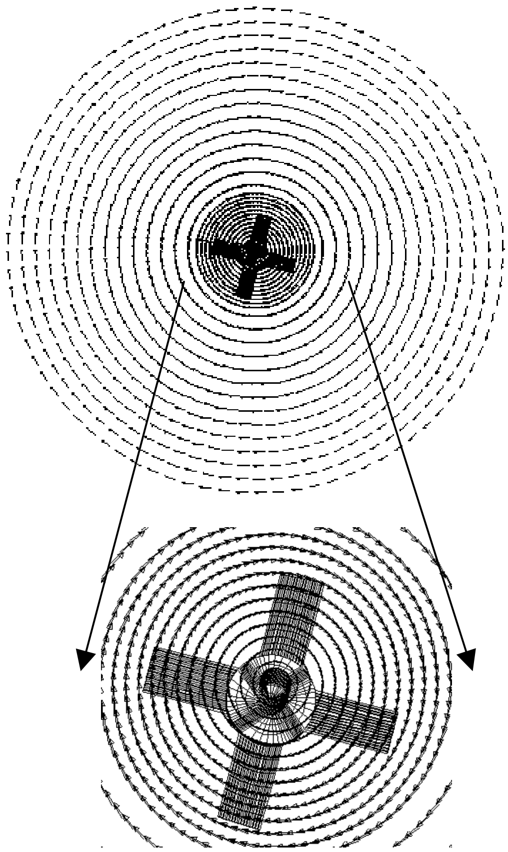

Figure 7: Flow profiles at top of kettle (No Baffles).

Figure 8 shows the velocity profile along a plane through the kettle close to an impeller blade. The magnitude of the velocity within the plane is small compared to that in the direction perpendicular to the plane. This, again, is due to rigid body motion being dominant in the flow for this design. This was also observed in the water model where polystyrene beads, whose density ratio to water is similar to that of dross and lead, were placed into the flow. Also, it could clearly be seen that beads within the water followed the flow profiles given by the CFD predictions.

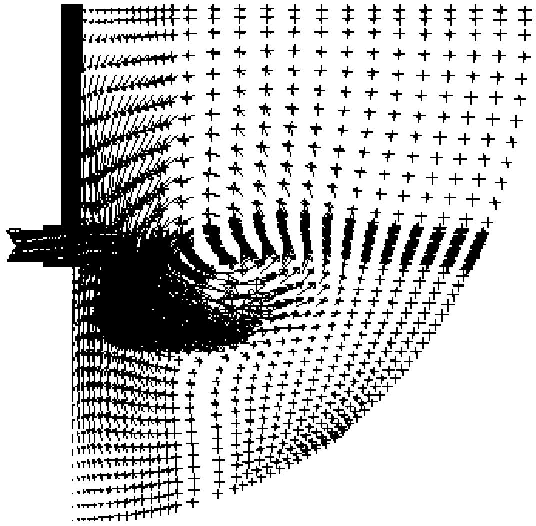

Figure 8: Flow profile in kettle (No Baffles).

Figures 9 and 10 show corresponding velocity profiles in a kettle that has two baffles. Clearly we can see that the flow profile on the top surface, Figure 9, is now moving into the shaft region, where the baffles are redirecting the flow into the center.

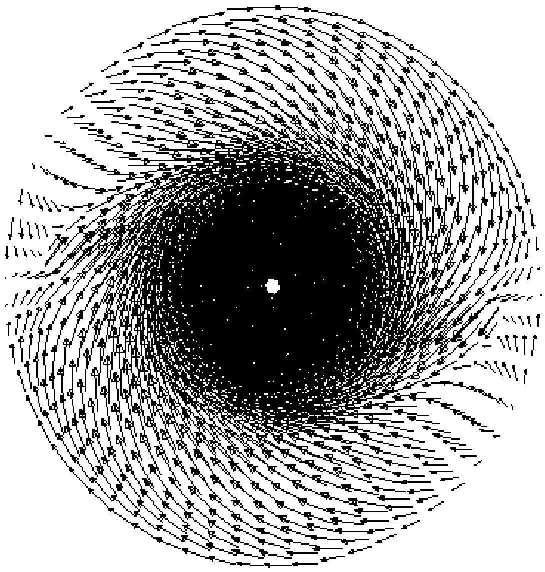

Figure 9: Velocity profile on bath surface (Baffles included).

Figure 10 shows the flow in the kettle along a plane close to the impeller. Here the flow is being directed down from the blade towards the lower kettle wall. A circulation profile appears to be present across the kettle.

During the water model tests, the motion of the plastic beads was observed for the baffled design. It was clear that the beads traveled throughout the kettle in a manner similar to that predicted by the CFD code.

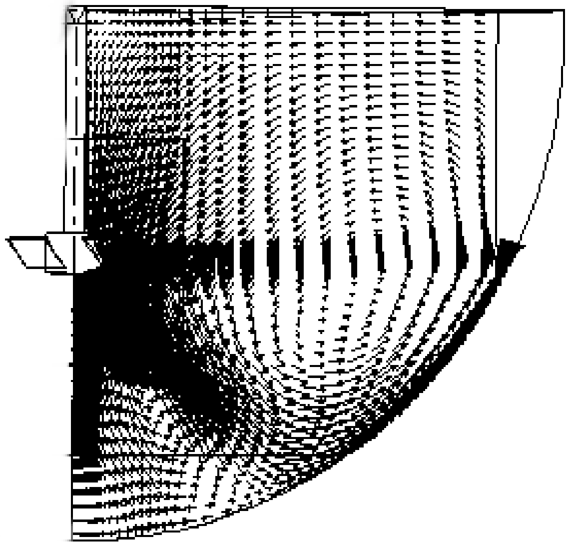

Figure 10: Flow profile in kettle (Baffles included).

Figure 11 shows the sampling points used to gather velocity data using the NORTEK probe. The CFD results for both the baffled and unbaffled case have been analyzed and velocity data at these points is compared with the probe readings.

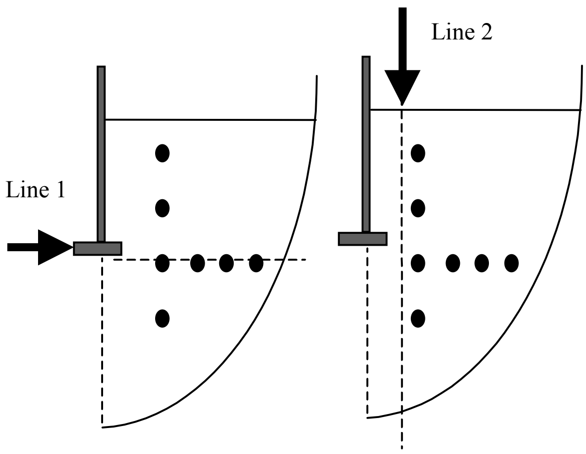

Figure 11: Sampling points.

Figures 12 and 13 compare the CFD results with the gathered probe data along lines (1) and (2) for the baffled kettle design. Figure 12 compares the CFD results for velocity along line (1) from the shaft to the kettle edge. Clearly at the shaft-impeller region the velocity is at its highest and then drops off sharply away from the blades. This region, up to 0.2m from the shaft, corresponds to the downward jet region observed in figure 10.

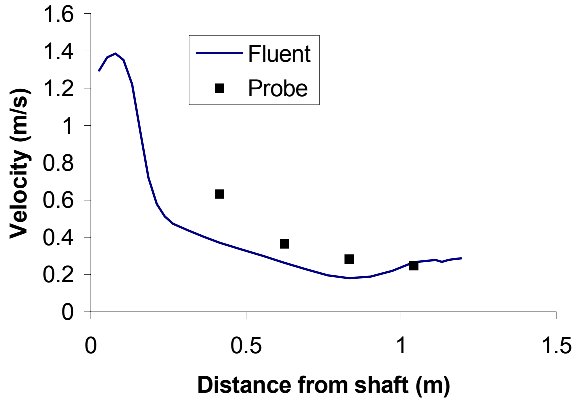

Figure 12: Velocity comparisons for sampling points on line (1) - (baffles included).

Comparisons between the probe velocity data and that predicted by CFD are very encouraging. Although no data was obtained close to the impeller shaft, the data away from the shaft compares well.

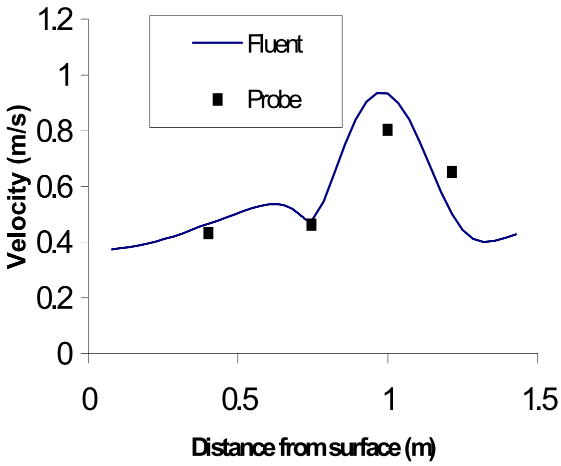

Figure 13: Velocity comparisons for sampling points on line (2) - (baffles included).

Figure 13 compares CFD predictions with gathered data for the set of sampling points along line (2). Again the comparisons are very encouraging. Both the CFD calculation and the probe readings capture the peak in velocity magnitude, below the impellers at approximately one-meter from the kettle surface. Also, the CFD predictions have picked up the flow velocities nearer the surface.

Figures 14 and 15 compare the CFD results with the gathered probe data along lines (1) and (2) for the water model without baffles. Although both the predicted and measured results are similar to those observed using baffles, there is some discrepancy between the data, especially close to the shaft and the bath surface. It is believed that this is due to surface swirl.

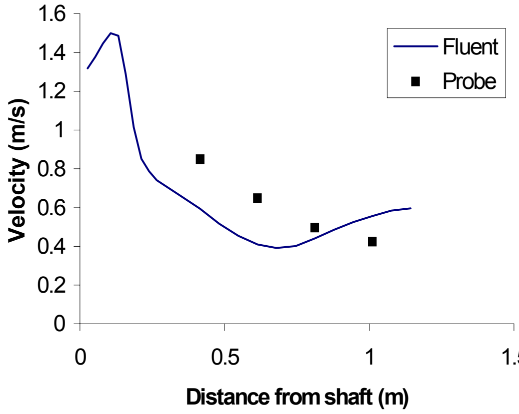

Figure 14: Velocity comparisons for sampling points on line (1) - (no baffles).

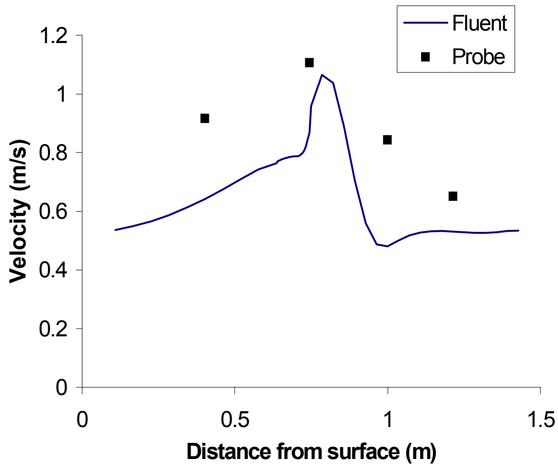

Figure 15: Velocity comparisons for sampling points on line (2) - (no baffles).

Lead Kettle Comparisons

A real lead-kettle design was used to obtain data from the lead mixing process. Three different tests are presented here, each using a four-bladed impeller where the blades are flat. Each test (T1, T2, T3) had an increase in shaft speed. Figure 16 shows the velocity profiles through the kettle along the plane of an impeller blade. As expected, this impeller design is producing a radial jet out from the blade and is forcing lead into the blade region from both above and below it, as seen in figure 17.

Figure 18 shows the comparison between gathered plant data and the CFD calculations for the velocity at the surface of the lead bath. The experimental data was gathered using the technique outlined above. Clearly we can see that for the lower shaft speeds (T3 and T2) the comparisons are very close, but for the higher speed (T1) a clear difference is present.

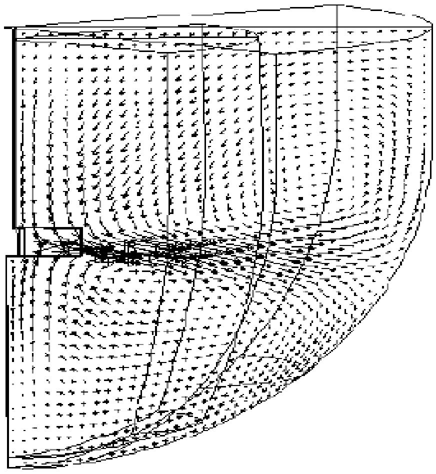

Figure 16: Velocity profile for real-lead kettle.

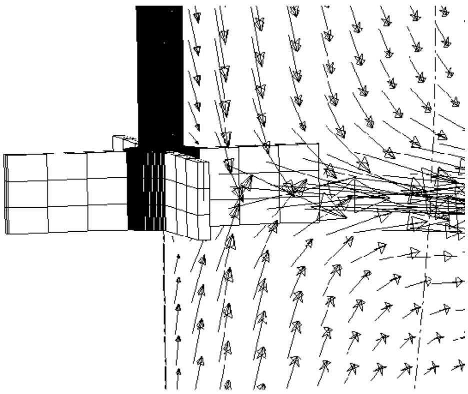

Figure 17: Velocity profile around impeller blades.

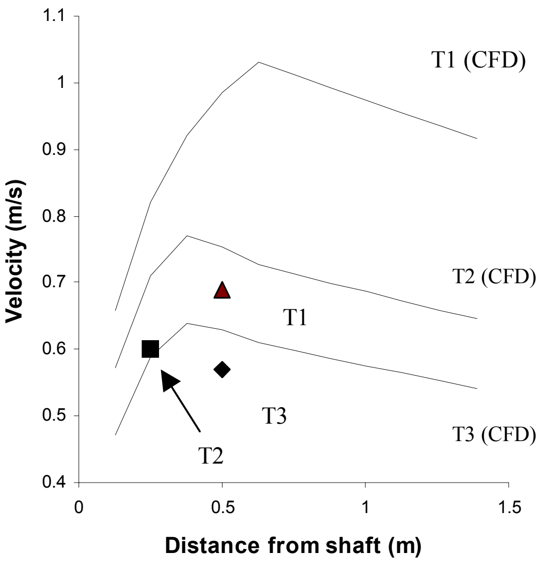

Figure 18: Comparisons between velocity data across surface of kettle.

CONCLUSION

A detailed modelling study of the lead refining process is now underway. Both physical and mathematical modelling technologies are being used to help understand key parameters in the kettle mixing process. Comparisons between CFD predictions and water model data for different flow regimes are encouraging. In future a dross layer will be added to the CFD model. Process conditions and impeller designs will be investigated further to optimise the process in terms of dross yield and mixing time.

REFERENCES

A BAKKER and HEA VAN DEN AKKER, “ Single Phase Flow in Stirred Tank Reactor”, *Trans IChemE*, 72, (1992) 583-593.

T R DAVEY ET-AL, “Flow Characteristics of Molten Lead”, *Symp. On Chem Engng in the Metallurgical Industries*, Instn Chem Engrs Pub. (1963) 105-117.

M F EDWARDS, in *Mixing in the Process Industries*, ed. N Harnby, M F Edwards and A W Nienow, Butterworths Pub., London (1985) 131-144.

FLUENT V4.5, Fluent Inc, Lebanon, NH (USA)

S KUMAR, C BAILEY, M PATEL, A.W PIPER and R FORSDICK, "Modelling the mixing of lead bullion during the refining process", in *Fluid Flow Phenomena in Metals Processing, Pub TMS 1999.*

NORTEK ACOUSTIC DOPPLER VELOCIMETER SYSTEMS - NORTEK AS, Norway.

CY PERNG and JY MURTHY, "A Moving-Deforming-Mesh Technique for Simulation of Flow in Mixing Tanks" *AICHE Symposium Series No 293, (1993), 37-39*

VV RANADE and SMS DOMMETRI, “Computational Snapshot of Flow Generated by Axial Impellers in Baffled Stirred Wessels”, *Trans IChemE*, 74A, (1996) 477-484.

ACKNOWLEGDMENTS

The authors would like to thank Britannia Refined Metals Ltd. for funding this project