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

EROSION PREDICTION IN SLURRY PIPELINE TEE-JUNCTIONS

Gary BROWN

Alcoa World Alumina Australia, Kwinana Refinery, Kwinana, Western Australia 6167

ABSTRACT

A significant erosion problem is found to occur on steel blanks located inside tee-junctions in a slurry pipeline system. The commercial CFD package CFX4 is used to predict the motion of caustic liquor and bauxite particles through a tee-junction using an Eulerian-Eulerian continuum approach in conjunction with the k-ε turbulence model.

Initial simulations assuming a uniform inlet flow are unable to demonstrate the cause of the observed erosion. However, subsequent modelling with a swirling inlet flow, based on a more thorough assessment of the upstream vessels and piping, results in the prediction of an accumulation of particles on the steel blank at the centre of a slow-moving vortex, the location of which is in excellent agreement with the observed wear on the plant. Furthermore, the predicted wear location is found to be insensitive to the assumed level of inlet swirl and the numerical scheme employed.

This result highlights the potential sensitivity of modelling results to inlet boundary condition assumptions and emphasises the need to adequately account for upstream influences when applying CFD techniques to the simulation of industrial flows.

NOMENCLATURE

C	inter-phase momentum transfer coefficient
C_D	drag coefficient
d_p	particle diameter
E	erosion rate
$\boldsymbol{g}$	gravitational acceleration
k_f	fluid turbulent kinetic energy
L_s	characteristic length
p	fluid pressure
r_f	fluid phase volume fraction
r_p	particulate phase volume fraction
Re_p	particle Reynolds number
S_t	Stokes number
$\boldsymbol{u}_f$	fluid velocity
$\boldsymbol{u}_p$	particle velocity
V_s	characteristic velocity
β	particle mass loading
ε_f	fluid turbulence dissipation rate
μ_{eff}	effective viscosity for fluid phase
μ_f	fluid dynamic viscosity
μ_{lp}	arbitrary laminar viscosity for particle phase
ρ_f	fluid density
ρ_p	particle density

INTRODUCTION

At Alcoa's Pinjarra alumina refinery steel blanks located inside tee-junctions in a slurry pipeline system are used to switch the flow between two possible paths (figure 1). The slurry being transported in this section of the plant consists of bauxite particles in a hot caustic soda solution. The bauxite particles have a high silica content and are hence highly abrasive.

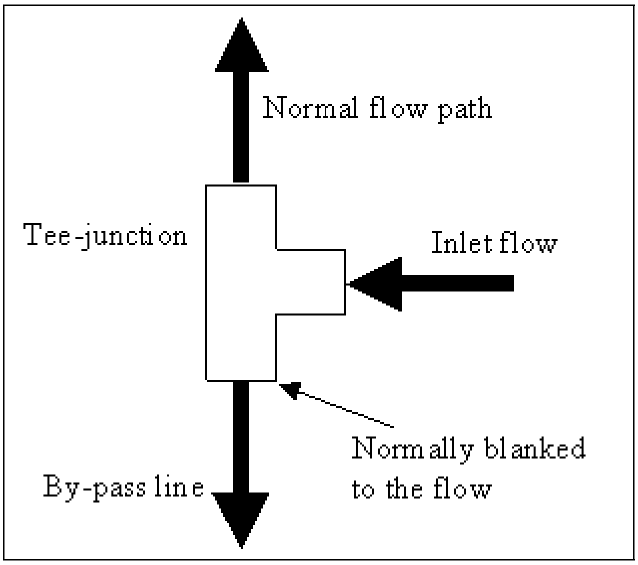

Figure 1: Slurry pipeline tee-junction

A significant erosion problem was found to occur on the steel blanks in the tee-junctions, when the blanks were located in the standard operating position (figure 1). It was found that a concentrated hole was being worn through these blanks inside 13 weeks of operation, allowing slurry into the by-pass piping (figure 2). The slurry would then stagnate and cool inside the by-pass piping, resulting in the creation of a hard scale that rendered the by-pass piping unusable.

The presence of such significant and highly localised erosion on the steel blanks was unexpected because the blanked end of the tee-junction was expected to be a region of near-stagnant flow. This lack of knowledge regarding the cause of the erosion made it difficult for plant engineers to develop a solution, other than to recommend the use of harder materials for the blanks. Consequently, a computational fluid dynamics (CFD) study was initiated to determine the cause of the erosion and to assist in the design of modifications to the piping system.

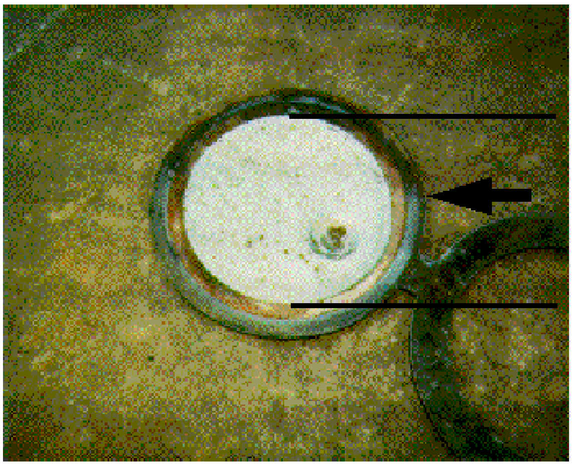

Figure 2: Hole in steel blank. Direction of inlet flow to tee-junction is indicated.

This paper describes the use of the commercial package CFX4.2 (AEA Technology, 1997) to predict the motion of caustic liquor and bauxite particles through a single tee-junction using an Eulerian-Eulerian continuum approach in conjunction with the k-ε turbulence model. In particular, the influence of the assumed inlet conditions to the tee-junction is discussed.

Single phase flow predictions are also conducted using a differential Reynolds stress turbulence model, in conjunction with a second order accurate differencing scheme (Van Leer), to confirm that the model results are qualitatively insensitive with respect to the turbulence model and differencing scheme selected.

PARTICLE TRANSPORT MODEL SELECTION

The three most common particle transport models, and those most prevalent in commercial CFD codes, are the Eulerian-Eulerian, Lagrangian and drift-flux models. All three of these models could potentially be applied to the simulation of a mineral processing slurry, depending on the exact slurry characteristics and the flow geometry in question. It is therefore worth reviewing the advantages and disadvantages of each model with respect to this application.

Eulerian-Eulerian models

Eulerian-Eulerian models are also generally referred to as multi-fluid or continuum models because the particles are treated as an additional continuous phase. An additional set of conservation equations is solved for the particulate phase and coupling between the two phases takes place through inter-phase transfer terms in the two sets of conservation equations.

This approach is ideally suited to the modelling of slurries with moderate to high particle concentrations, where two-way particle-fluid coupling and possibly particle-particle interactions are important, because source terms can readily be formulated in terms of spatial gradients in phase velocities and volume fractions (Andrews and O'Rourke, 1996). These features have been exploited, for example, in the modelling of thickener vessels used in the mineral processing industry (Kahane *et al.*, 1997).

The main limitation of the Eulerian-Eulerian approach is in its application to systems with a large particle size distribution. Additional continuity and momentum equations are required for every additional particle size simulated, hence significantly increasing the complexity of the model formulation and the computational overhead. As a result, the Eulerian-Eulerian approach is usually applied in situations where the important features of the flow can be discerned through the simulation of a single representative particle size.

Lagrangian models

In the Lagrangian approach Eulerian continuum equations are still solved for the fluid phase (the carrier liquid in the case of a slurry) but Newtonian equations of motion are solved to determine the trajectories of individual particles (or groups of particles). Each particle can have different properties (size, shape, density, initial conditions) thus allowing the simulation of slurries involving large particle size distributions or different ore types.

In flows where the particle loading is high enough that significant two-way particle-fluid coupling would be expected, the particle trajectories can be evaluated in a coupled loop with the continuous phase solution. However, achieving realistic coupling between the particle and fluid phases requires adequate resolution of the spatial distribution of the particles, which in turn requires the calculation of a large number of trajectories. This makes use of the Lagrangian approach computationally expensive for flows in which two-way coupling must be considered. In addition, in Lagrangian simulations the volume fraction occupied by the particle phase is generally ignored in the solution for the continuous phase. This assumption would obviously be invalid at anything other than dilute particle loadings.

These features of the Lagrangian approach make it ideally suited to cases in which the determination of particle classification is important and in which the solids loading is low, thus allowing two-way coupling and the volume occupied by the particles to be neglected and the particle trajectories to be calculated as a post-process. A recent example in the mineral processing industry has been the prediction of spiral concentrator performance (Matthews *et al.*, 1998).

Drift-flux models

Drift-flux or algebraic slip models involve the solution of Eulerian continuum equations for a single fluid phase which exhibits a variable density based upon the local particle volume fraction summed across all particle sizes (ie. the fluid density is the effective slurry density). The particles are assumed to be continuously slipping with respect to this carrier fluid at a constant velocity due to gravitational and/or centrifugal forces and the distribution of particles is obtained through the solution of a single scalar transport equation for the volume fraction of each particle size considered.

This technique has obvious advantages in cases where the Eulerian-Eulerian approach is computationally too prohibitive due to the need to consider multiple particle sizes and yet the Lagrangian technique is also unsuitable because the particle concentration is high enough to significantly influence the fluid flow. However, because of the need to calculate a fixed particle slip velocity the use of drift-flux models is generally limited to geometries in which either gravitational and/or centrifugal forces dominate, such as in gravity settling vessels (Li *et al.*,

1998) and hydrocyclones (Pericleous, 1987 and Zughbi *et al.*, 1991).

Particle-fluid coupling

Selection of the correct particle transport model for a particular application can be assisted by first calculating the particle mass loading, β, and the Stokes number, S_t.

The particle mass loading is expressed as:

$$\beta = \frac{\text{particulate mass per unit volume of flow}}{\text{fluid mass per unit volume of flow}} = \frac{r_p \rho_p}{r_f \rho_f} \tag{1}$$

where r is a volume fraction, ρ is a density and the subscripts p and f refer to the particle and fluid phases respectively. Significant two-way particle-fluid coupling is generally expected for particle mass loadings greater than 0.2 and values greater than 0.6 indicate that significant particle-particle interactions are likely in at least some parts of the flow domain.

The degree to which the particle motion is tied to the fluid motion can be determined through evaluation of the Stokes number. This is defined as the ratio of the particle response time due to viscous drag to a characteristic turbulent eddy time in the carrier fluid. This can be expressed as:

$$S_t = \frac{\rho_p d_p^2 V_s}{18 \mu_f L_s} \tag{2}$$

where d_p is the particle diameter, μ_f is the dynamic viscosity of the carrier fluid and V_s and L_s are characteristic velocity and length scales in the flow.

For large values, S_t>2.0, the particulate flow is highly inertial and, in a confined geometry, would be dominated by particle-wall interactions, whereas for values less than 0.25 the effect of particle-wall interactions on the particle flow is essentially negligible because the particles are more tightly coupled to the fluid through viscous drag (Tu and Fletcher, 1996). At Stokes numbers below 0.05 the particles and carrier fluid are strongly coupled and the particles would be expected to approximately follow the fluid flow. For very small values, say S_t<<0.01, particle inertia is essentially negligible and the particles would be expected to respond almost instantaneously to any changes in the fluid flow.

Model selection

For the tee-junction flow to be evaluated in the current study the application of a drift-flux model was not considered because the geometry is such that the particle motion would not be dominated by either gravity settling or centrifugal separation.

The particle mass loading at inlet was calculated to be 0.175 which indicated that it may have been possible to neglect two-way particle-fluid coupling. However, some particle settling was expected to occur in the blanked end of the tee which would increase the particle mass loading in this area to a level where two-way coupling might influence the fluid flow. Furthermore, prediction of erosion on the steel blank would obviously require adequate resolution of the particle distribution in an area which is off the mean flow path. Therefore, the use of a Lagrangian model would be computationally expensive due to the need to calculate a large number of particle trajectories in a coupled solution.

The Stokes number based on the inlet velocity, pipe diameter and mean particle size was calculated to be 0.026, which indicated that particle-wall interactions would have a negligible effect on the particle flow and hence that no special treatment of the particle boundary conditions was required.

In consideration of the above analysis an Eulerian-Eulerian model was selected for the tee-junction study.

EROSION MODELLING

Following the concepts established by Finnie and Bitter, erosion of a solid surface due to particle impacts can be considered to be due to two separate mechanisms, namely deformation wear and cutting wear (Finnie, 1960 and Bitter, 1963).

Deformation wear occurs when repeated particle impacts at high impact angles plastically deform the surface layers of the material, eventually causing material loss through surface fragmentation. Cutting wear occurs due to particle impacts at small angles, with a scratch or cut being formed on the surface if the shear strength of the material is exceeded.

The total erosion rate at a particular point on a surface is found by summing the contributions due to the deformation and cutting mechanisms and depends on the properties of the material, with deformation wear being more significant for hard, brittle materials and cutting wear being more significant for softer, ductile materials. For standard commercial grade steels, as used in the tee-junction in this study, peak erosion rates have been measured to occur at impact angles of 25-30°, indicating that cutting wear dominates (Bitter, 1963).

The other critical factor affecting wear is the particle impact velocity, with both cutting and deformation wear being proportional to impact velocity raised to a power n determined through physical tests. In general n is found to vary between 2.0 and 3.0 depending on both the surface and particle materials.

These concepts were embodied by Finnie in a simplified erosion model relating erosion rate to particle mass flux, impact angle and impact velocity (Finnie, 1960). This model or its variants have been used successfully in many CFD erosion studies, for example in the examination of fluidised beds using an Eulerian-Eulerian model (Achim *et al.*, 1995).

An erosion model has not been implemented in the current study because the cause of the erosion was found to be evident from an examination of the particle distribution and flow patterns within the tee-junction, as presented in the results below. However, the author intends to extend this study to include the implementation of an erosion model in the near future.

EULERIAN-EULERIAN (MULTI-FLUID) MODEL DESCRIPTION

Governing Equations

A two-phase Eulerian-Eulerian model has been established for the liquid phase (caustic soda solution) and a single representative bauxite particle size, in this case selected to be particles of 150 micron diameter. The full solids mass loading is assumed to exist at this particle size.

Turbulent dispersion of the particles is neglected and the particle mass loading in the majority of the flow domain is considered to be low enough to allow the effect of particles on fluid phase turbulence and particle-particle interactions to also be neglected. As a result of this last assumption the particle phase is declared within CFX to be laminar and is attributed a small dynamic viscosity. This has the effect of making the diffusion term in the particle phase momentum equation negligible.

As a result of these assumptions and for steady flow the required continuum equations for conservation of mass and momentum are:

$$\nabla.\left(r_f \rho_f \mathbf{u}_f\right) = 0 \tag{3}$$

$$\nabla.\left(r_p \rho_p \mathbf{u}_p\right) = 0 \tag{4}$$

$$\begin{aligned}&\nabla.\left(r_f \rho_f \mathbf{u}_f \mathbf{u}_f\right) - \nabla.\left\{r_f \mu_{eff}\left[\nabla\mathbf{u} + (\nabla\mathbf{u})^T\right]\right\}_f \\ &= -r_f \nabla p + C\left(\mathbf{u}_p - \mathbf{u}_f\right) + r_f \rho_f \mathbf{g}\end{aligned} \tag{5}$$

$$\begin{aligned}&\nabla.\left(r_p \rho_p \mathbf{u}_p \mathbf{u}_p\right) - \nabla.\left\{r_p \mu_{lp}\left[\nabla\mathbf{u} + (\nabla\mathbf{u})^T\right]\right\}_p \\ &= -r_p \nabla p + C\left(\mathbf{u}_f - \mathbf{u}_p\right) + r_p \rho_p \mathbf{g}\end{aligned} \tag{6}$$

where it should be noted that μ_{lp} is an arbitrarily small laminar viscosity for the particle phase.

In addition, turbulence closure in the fluid phase is achieved through solution of the standard k-ε model, with the k and ε equations taking the form:

$$\nabla.\left(r_f \rho_f \mathbf{u}_f \phi\right) - \nabla.\left(r_f \frac{\mu_{eff}}{\sigma_\phi} \nabla\phi\right) = r_f S_\phi \tag{7}$$

where ϕ represents either k or ε, σ_ϕ is the turbulent diffusivity of ϕ and S_ϕ is a source term.

The inter-phase momentum transfer coefficient C is calculated through consideration of fluid-particle drag. The particle drag coefficient is given by:

$$C_D = \frac{24}{\mathrm{Re}_p}\left(1 + 0.15\,\mathrm{Re}_p^{0.687}\right) \quad 0 < \mathrm{Re}_p \leq 1000 \tag{8}$$

with the particle Reynolds number defined as:

$$\mathrm{Re}_p = \frac{r_f \rho_f d_p \left|\mathbf{u}_p - \mathbf{u}_f\right|}{\mu_f} \tag{9}$$

Boundary Conditions

All variables are defined at the tee-junction inlet. The particles are also assumed to be uniformly distributed and, due to the low Stokes number, the particle velocity distribution is assumed to be identical to that for the fluid phase. A zero gradient condition is applied at the outlet.

Standard no-slip wall functions are applied at all solid surfaces for both the fluid and particle phases. Use of no-slip wall functions for the particle phase is justified in this case on the basis of the low Stokes number (0.026) which indicates that there is a strong coupling between the particles and the fluid and hence that particle-wall rebound characteristics will have a negligible impact on the particle flow.

Computational Domain and Numerical Procedure

The tee-junction geometry is discretised using a block-structured non-orthogonal Cartesian mesh. Due to the circular pipe sections a 5-block mesh structure is adopted to reduce the non-orthogonality of the mesh elements. The computational mesh is shown in figure 3.

The multi-phase conservation equations for mass, momentum and fluid turbulence are solved within the commercial code CFX4 using a finite volume technique based on the SIMPLEC algorithm. Inter-phase coupling is achieved using Spalding's Inter-Phase Slip Algorithm (IPSA). Convection terms in the momentum equations are discretised using the first order hybrid scheme.

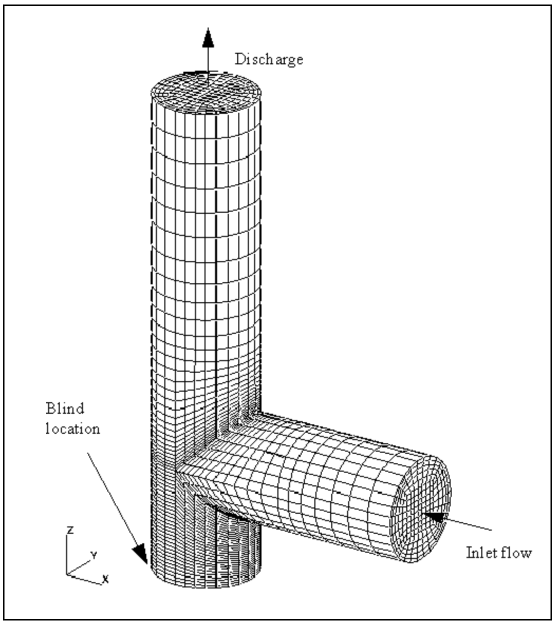

Figure 3: Computational domain.

EULERIAN-EULERIAN MODEL RESULTS

Initial simulations of the tee-junction were undertaken using an assumption of a uniform inlet flow and particle distribution. The resulting flow patterns and particle distribution are shown in figure 4.

It can be seen that a slow moving recirculation is created in the blanked end of the tee as the main flow moves from the inlet leg to the discharge leg. Particles entrained in this recirculation from the main flow move down the back wall of the tee and across the face of the blank, before being deposited at the bottom of the front wall of the blanked area - the vertical component of fluid velocity next to the wall being insufficient to suspend the particles. By comparison with figure 2 it can be seen that this result is inconsistent with the highly localised and asymmetric erosion observed on the plant.

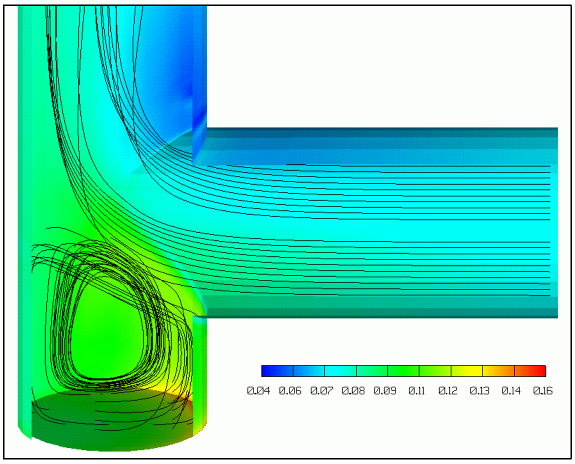

Figure 4: Particle concentration (v/v) and fluid phase streamlines. Multi-phase model with uniform inlet flow.

In light of this result a more thorough assessment of the piping and vessel layout upstream of the tee-junction was undertaken. It was subsequently concluded that there was significant potential for the inlet flow to be swirling, because the slurry is drawn from the upstream vessel through a single centrally-located underflow pipe.

Further simulations were conducted with a swirl velocity component added to both the fluid and particle phases at the inlet to the computational model. Solid body rotation was assumed, with the peak tangential velocity occurring at the pipe wall. The particle distribution at inlet was still assumed to be uniform. In the absence of any data regarding the potential level of swirl in the flow an initial simulation was conducted with the peak tangential velocity at inlet set equal to the inlet axial velocity. The results achieved are illustrated in figures 5 and 6.

In figure 5 streamlines for the fluid phase have been initiated from several positions at the tee-junction inlet and also from several positions on the surface of the steel blank. It can be seen that the rotation of the inlet flow generates a vortex in the blanked region of the tee, with the vortex centred about a point remarkably close to the site of the observed erosion on the plant (figure 2). In addition, a plot of particle concentration reveals an accumulation of particles on the surface of the steel blank at the same point (fluid phase streamlines in the blanked end of the tee have been removed in figure 6 for clarity).

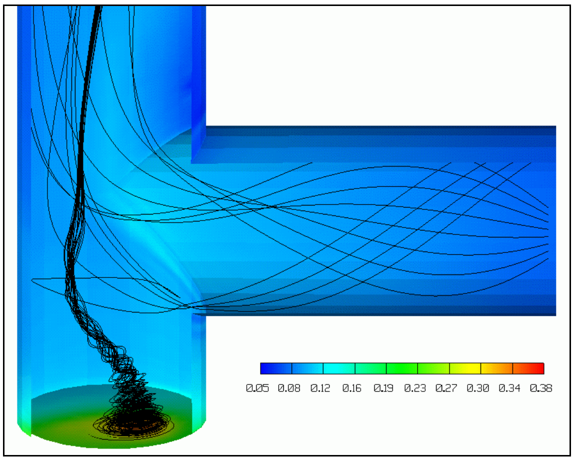

Figure 5: Particle concentration (v/v) and fluid phase streamlines. Multi-phase model with swirling inlet flow.

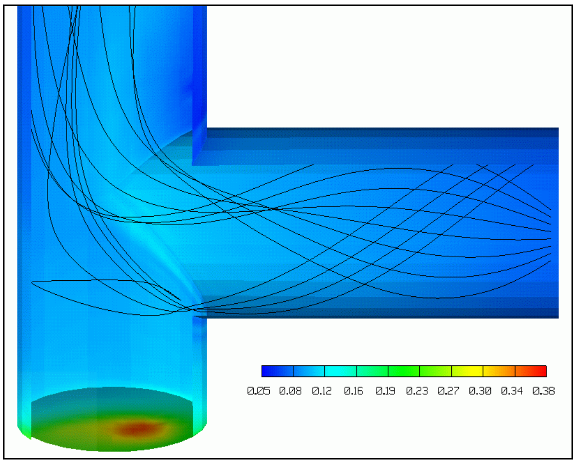

Figure 6: Particle concentration (v/v) and fluid phase streamlines (streamlines in blanked end of tee removed for clarity). Multi-phase model with swirling inlet flow.

Further analysis of particle phase streamlines revealed that particles entering the blanked end of the tee are eventually entrained into the centre of the vortex before being drawn up the vortex core. This suggests that the highly localised erosion seen on the steel blank in the plant is due to repeated particle impacts at highly acute angles as particles are entrained into the centre of the vortex. Given the small bed of solids at the centre of the vortex there may also be the potential for erosion due to friction between particles sliding in the bed and the surface of the blank

Given the lack of data regarding the level of swirl in the inlet flow on the plant, simulations were also conducted with the peak tangential velocity at inlet set to 1.5x and 0.5x the inlet axial velocity. In both cases a vortex was found to form in the blanked end of the tee, centred about the same point as in the initial simulation, and with an accumulation of solids on the steel blank at the core of the vortex. As expected, there are quantitative variations in the results, with rotational velocities in the vortex increasing and the solids concentration in the core decreasing as the inlet swirl increases. However, these results show importantly that the predicted flow is qualitatively insensitive to the assumed level of inlet swirl.

SINGLE PHASE DIFFERENTIAL STRESS RESULTS

Fluid phase streamlines were found to be almost identical in the k-ε multi-phase runs, to those in single phase k-ε runs. This indicates that the relatively low particle mass loading in the majority of the flow (β<0.2), and even the higher mass loadings in the blanked end of the tee, do not result in any significant alteration to the fluid flow field due to two-way coupling. It was therefore concluded that it would be sufficient to use a single phase model to check the sensitivity of the results to the numerical scheme selected.

A single phase simulation was conducted for the swirling inlet flow case using a Differential Reynolds Stress Model (DSM) in conjunction with the second-order accurate Van Leer differencing scheme, on the basis that this has been shown to give significantly improved accuracy for swirling flows on non-orthogonal Cartesian grids when compared to use of the standard k-ε turbulence model and hybrid differencing (Shore et al., 1995).

The predicted flow patterns are shown in figure 7. Comparison with the k-ε results in figure 5 shows that there is some variation in the structure and location of the vortex, but that the overall nature of the flow does not change significantly. On the basis of this result, highly localised erosion would still be expected at a site in close agreement with the multi-phase k-ε results and the observed erosion on the plant. This demonstrates that the results are relatively insensitive to the exact numerical scheme selected.

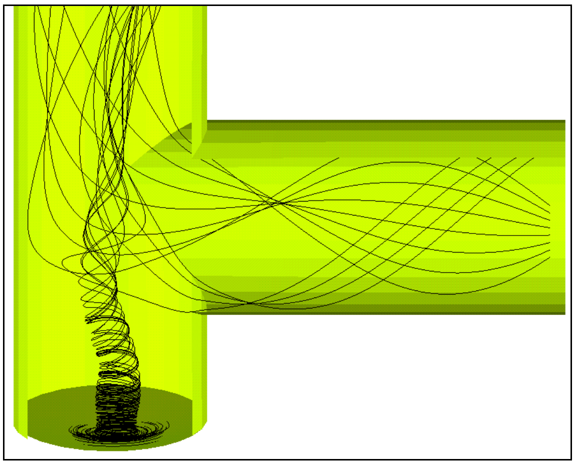

Figure 7: Fluid phase streamlines. Single phase model with swirling inlet flow (DSM + Van Leer).

CONCLUSIONS

The commercial CFD code CFX4 has been used to investigate the cause of highly localised and asymmetric erosion found to occur on steel blanks within slurry pipeline tee-junctions. The motion of caustic liquor and bauxite particles through a tee-junction has been predicted using an Eulerian-Eulerian multi-phase approach in conjunction with a k-ε turbulence model.

Simulations with an assumed uniform inlet flow were unable to identify the cause of the erosion. However, simulations with a swirling inlet flow, based on a more thorough assessment of the upstream vessels, showed an accumulation of particles on the steel blank at the centre of a slow-moving vortex, the location of which is in excellent agreement with the observed wear on the plant. This result was found to be qualitatively insensitive to the assumed level of inlet swirl.

Comparison of the multi-phase results with single phase predictions has shown that the particles do not significantly alter the fluid flow in this case due to the relatively low solids mass loading in the majority of the flow domain. On this basis, a single phase simulation has also been conducted using a Differential Reynolds Stress Model in conjunction with a second-order accurate convective differencing scheme. The results obtained demonstrate that the predicted erosion site is also insensitive to the numerical scheme employed.

These results demonstrate how CFD techniques can be used to significant benefit in the prediction of industrial erosion problems. However, the results also demonstrate how deceptively complex flows can be established inside simple geometries due to non-uniformities in the inlet flow, thus highlighting the need to adequately account for upstream influences when applying CFD techniques to the simulation of industrial flows.

REFERENCES

ACHIM, D., EASTON, A.K., PERRY, J.H. and SCHWARZ, M.P. (1995), "Mathematical modelling of erosion in a fluidised bed", *Proc. 12th Australian Fluid Mech. Conf.*, Sydney, Australia, December 10-15.

AEA TECHNOLOGY (1997), "CFX-4.2 Flow solver user guide", AEA Technology plc.

ANDREWS, M.J. and O'ROUKE, P.J. (1996), "The multiphase particle-in-cell (MP-PIC) method for dense particulate flows", *Int. J. Multiphase Flow*, **22**, 379-402.

BITTER, J.G.A. (1963), "A study of erosion phenomena: Parts 1 and 2, *Wear*, **6**, 5-21 & 169-190.

FINNIE, I. (1960), "Erosion of surfaces by solid particles", *Wear*, **3**, 87-103.

KAHANE, R.B., SCHWARZ, M.P. and JOHNSTON, R.M. (1997), "Residue thickener modelling at Worsley Alumina", *Proc. Int. Conf. on CFD in Mineral & Metal Processing and Power Generation,* CSIRO*,* Melbourne, Australia, July 3-4.

LI, Y., RUDMAN, M. and BROWN, G. (1998), "Particle transport in a bottom-feed separation vessel", *Appl. Math. Modelling*, **22**, 1023-1036.

MATHEWS, B.W., FLETCHER, C.A.J. and PARTRIDGE, A.C. (1998), "Computational simulation of fluid and dilute particulate flows on spiral concentrators", *Appl. Math. Modelling*, **22**, 965-979.

PERICLEOUS, K.A. (1987), "Mathematical simulation of hydrocyclones", *Appl. Math. Modelling*, **11**, 242-255.

TU, J.Y. and FLETCHER, C.A.J. (1996), "Eulerian modelling of dilute particle-laden gas flows past tubes", *Computational Fluid Dynamics Journal*, **5**, 1-25.

SHORE, N.A., HAYNES, B.S., FLETCHER, D.F. and SOLA, A.A. (1995) "Numerical aspects of swirl flow computation", *Computational Techniques and Applications: CTAC95*, Melbourne, Australia, July 3-7.

ZUGHBI, H.D., SCHWARZ, M.P., TURNER, W.J. and HUTTON, W. (1991), "Numerical and experimental investigations of wear in heavy medium cyclones", *Minerals Engineering*, **4**, 245-262.