Third International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
10-12 December 2003

CFD SIMULATIONS OF GRAVITY SLUICES

DR MS BRENNAN

The Julius Kruttschnitt Mineral Research Centre, The University of Queensland, Isles Rd, Indooroopilly, Queensland, 4068, Australia

ABSTRACT

Centrifugal separators are widely used in the mineral processing industry to concentrate fine dense particles, such as fine gold. A 1-g analogue to centrifugal separators is the simple gravity sluice with a much simpler geometry. CFD simulations of gravity sluices using FLUENT have been completed as a precursor to modelling enhanced centrifugal separators such as the Knelson and Falcon devices. The air/water interface was modelled with the Volume of Fluid (VOF) and Algebraic Slip Mixture (ASM) models and two equation turbulence closures were used. Prediction of the free surface is heavily influenced by the quality of the mesh in the entrance region and the ASM gives better prediction of the free surface than does the VOF model. Prediction of velocity profiles is influenced by the near wall mesh. The k-omega model gives the best velocity profile prediction, however all two equation models do not properly model the anisotropy of the turbulence near the free surface and better turbulence closures are needed to model this aspect of the flow.

NOMENCLATURE

g gravitational acceleration – m.s^{-2}
$\mathbf{u}$, u velocity – m.s^{-1}
S source of phase k – kg.m^{-3}.s^{-1}
y distance from bottom of sluice - m

α phase volume fraction
θ sluice inclination
ρ density – kg.m^{-3}
τ stress tensor
μ dynamic viscosity

Subscripts
dm diffusion stress mixture
f liquid phase
km phase k relative to mixture
m mixture
tm turbulent stress mixture
k phase k
x stream-wise along sluice
y span-wise across sluice

INTRODUCTION

Gravity sluices are an inclined parallel sided channel. Their simple geometry makes generation of the CFD mesh straight forward, however several characteristics of the flow make CFD modelling of the devices complex. The flow is an example of gravity driven open channel flow, but the upper free surface of the liquid is located in the log layer. Thus the flow, while turbulent, is entirely within the boundary layer, and is also influenced by free surface effects.

The velocity profile in fully developed constant shear boundary layers obeys what is commonly called the Law of the Wall (Hunt 1964). In gravity sluices the shear stress across the flow is not constant but is a function of the weight of liquid above any point and is given by the following relationship, if momentum transfer to the air above the free surface is neglected:

$$\tau_{xy} = \rho_f g \sin\theta (y_s - y) \quad (1)$$

Since the shear stress is a maximum at the wall and drops to zero as the free surface is approached, one would expect negative deviation from the Law of the Wall behaviour with a flattening of the velocity profile as the free surface is approached. However experimental data for gravity driven open channel flows presented by Nezu and Rodi (1986) shows that this is not the case and in fact the Law of the Wall is still obeyed right up the free surface even when the free surface is in the log layer. Indeed the velocity profiles still show the normal positive deviation from the Law of the Wall in experiments where the free surface is located in the velocity defect layer.

Nezu and Rodi (1986) analysed their data using a mixing length model but used the distance to the upper free surface as well the distance to the wall to calculate the mixing length. This fitted their data well but implies that although the free surface is "free" in the sense of shear, it still acts as a scale determining boundary for the turbulence.

DNS studies of a gravity driven, open channel flow of this type by Nakayma and Yokojima (2001) predict that the Law of the Wall is obeyed up to the free surface. In this study the free surface was at a y^+ of 180. Inspection of Nakayma and Yokojima's (2001) data for turbulence statistics shows that the Reynolds shear stress $\overline{u'_x u'_y}$ obeys the expected deviation from the normal stress $\overline{(u'_x)^2}$ in the region of the wall, but also deviates by dropping to zero as the free surface is approached, while the normal stress remains positive. The $\overline{(u'_y)^2}$ stress also drops to zero as the free surface is approached.

Several authors have since attempted to model this type of flow using CFD. Prinos and Zeros (1995) used low Reynolds number k-epsilon models and obtained good results. Craft et al (1995) applied a non-linear eddy viscosity model to open channel flows and Craft and

Launder (1996) have applied a second moment closure which is a variant of the TCL model to open channel flows.

In this work we report on results which compare the performance of Fluent's realizable k-epsilon model (RKE) (Shih et. al., 1995), the k-omega model (Wilcox 1988, 1998) and the Fluent implementation of the Reynolds Stress model (RSM) for this type of flow when used in conjunction with the Volume of Fluid Model (VOF) (Hirt and Nichols, 1981) and the Algebraic Slip Mixture model (ASM) (Manninnen and Taivassalo, 1996) for predicted the velocity profiles in sluices. The results are compared to Nezu and Rodi's (1986) data for the same hydraulic Reynolds number and to experimental flow split data obtained at the JKMRC (Majumder 2002).

MODEL DESCRIPTION

Mathematical formulation

The aim was to produce a predictive hydrodynamic model for the gravity sluice, which could be extended to modelling multiphase flows and predict concentration and classification. The approach used was to model the position of the free surface using the VOF and ASM models. Both of these models are basic multiphase models, which solve the Reynolds Averaged Navier Stokes and Continuity equations for the mixture:

$$\frac{\partial}{\partial t}\left(\bar{\rho}_m \bar{\mathbf{u}}_m\right)+\nabla\cdot\left(\bar{\rho}_m \bar{\mathbf{u}}_m \bar{\mathbf{u}}_m\right)= -\nabla P_k+\nabla\cdot\left(\boldsymbol{\tau}_m+\boldsymbol{\tau}_{tm}+\boldsymbol{\tau}_{dm}\right)+\bar{\rho}_m \mathbf{g} \qquad (2)$$

$$\frac{\partial}{\partial t}\bar{\rho}_m+\nabla\cdot\bar{\rho}_m \bar{\mathbf{u}}_{\mathbf{m}}=0 \qquad (3)$$

The VOF model (Hirt and Nichols, 1981) is specifically designed to track the position of a free surface between two discrete phases and solves an additional continuity equation for the additional phase:

$$\frac{\partial}{\partial t}\alpha_k+\bar{\mathbf{u}}_{\mathbf{m}}\nabla\cdot\alpha_k=\frac{S_k}{\rho_k} \qquad (4)$$

The Fluent implementation of the VOF model has two interpolation schemes (eg, the Donor Acceptor and Geo-reconstruct schemes) which are designed to improve the prediction of the free surface.

The ASM model (Manninen and Taivasslallo, 1996) is primarily intended to model multiphase flows where dispersed phases are mixed with a continuous fluid phase, and it also solves an additional continuity equation for the dispersed phase:

$$\frac{\partial}{\partial t}\left(\rho_k\alpha_k\right)+\nabla\cdot\left(\alpha_k\rho_k\bar{\mathbf{u}}_{\mathbf{m}}\right)=-\nabla\cdot\left(\alpha_k\rho_k\bar{\mathbf{u}}_{km}\right) \qquad (5)$$

The ASM phase continuity equation differs from that for VOF model in it has an additional term for the drift velocity for the phase relative to that of the mixture. The drift velocity is determined using a drag calculation which assumes that the interphase momentum transfer is at local equilibrium. Thus the ASM model can model phase segregation under buoyancy or body forces. It will also track the position of a free surface in flows where the phases segregate totally.

The turbulent stresses in the mixture momentum equation (2) can be modelled by any of the standard turbulence closures. We have investigated the Realizable k-epsilon model (Shih et al, 1995), the k-omega model (Wilcox, 1988, 1998) and the Fluent implementation of the Reynolds Stress Model.

Mesh

The laboratory sluice (Majumder, 2002) was a 1.5m long channel with a maximum flow depth of 0.06m and was inclined at an angle of 17.5^o. A flow splitter was located at the discharge of the channel and the splitter height was adjustable between 0.5 and 3 mm. The sluice was 0.37m wide.

The simulations were carried out using Fluent 6 with two dimensional meshes, which were generated using Gambit. A mesh was generated for the full flow depth, the splitter and, inner and outer launders. As the liquid flow was thin, the meshes were graded so that most of the mesh points were in the liquid region with a coarser mesh in the air region.

As the flow was wall dominated, the mesh extended into the viscous sub layer; such that $y^+ \sim$ 1-3 in the wall bounded mesh points and this was so that enhanced wall functions could be used with the RKE and RSM models. The k-omega model can model boundary layer flows into the viscous sub-layer and only requires a wall function in the wall bounded grid point.

RESULTS

The initial studies using the VOF model predicted a free surface that was excessively diffuse. The Geo-reconstruct and the Donor Acceptor interpolation schemes which are available in Fluent to deal with this problem proved to be consistently unstable numerically, so the problem of sharpening up the free surface was addressed using grid refinement.

It was found that the diffuseness of the free surface started at the entrance to the sluice as the water level dropped from the feed inlet boundary to the height for a fully developed flow and the diffuseness then persisted to the end of the channel. By introducing more mesh points at the feed end of the grid using Fluent's grid adaptation facilities, the diffuseness was significantly reduced.

The "Adapt to Boundary" function was used where the region of the grid to be refined was selected based on a number of grid points normal to a chosen boundary, which in this case was the feed inlet boundary condition

Figure 1a shows the contours of volume fraction at the feed end of the sluice for the initial grid. Figure 1b shows contours of volume fraction after adapting the grid using boundary adaptation of 15 points from the feed, whilst Figure 1c shows the results from a further adaptation of 20 points. As can be seen the free surface is much sharper for the adapted grids. All these results used the basic VOF model with Shear Stress k-omega model and used a feed flow rate of 0.917 kg.s^{-1} (for 0.37m width).

Subsequently the ASM model was used to predict the position of the free surface. While the free surface prediction at the feed end of the sluice was similar to that predicted by the VOF model, the drift velocity introduced into the water continuity equation by treating water as a dispersed phase had the effect of sharpening up the prediction of the free surface such that the free surface extended across only two grid points at the discharge from the sluice. These results are shown in Figure 2. Note that in Figure 2, the height is expressed as y^+ using the wall shear stress as calculated using the modelled free surface position. The ASM model was used in all subsequent simulations reported here.

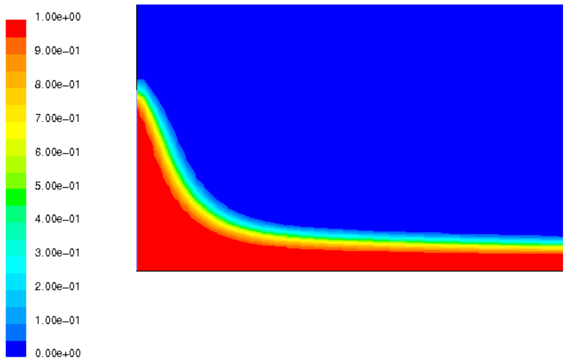

Fig 1a

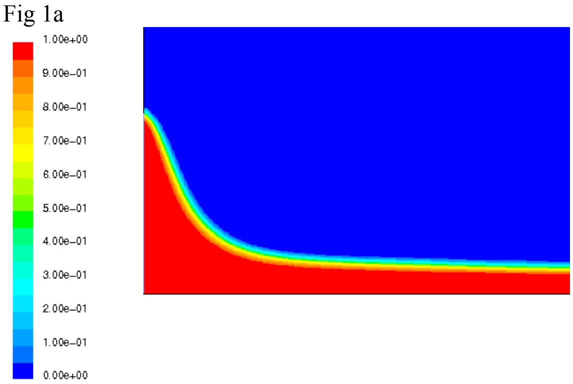

Fig 1b

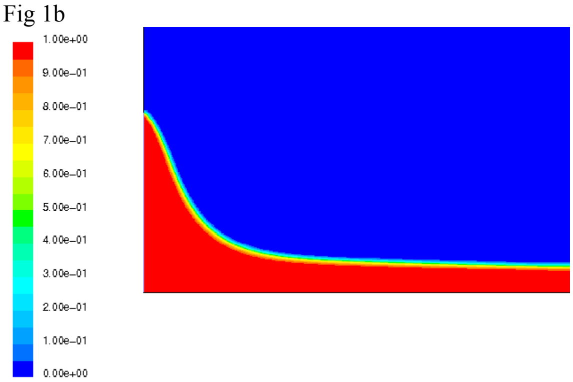

Fig 1c

Figure 1 – Influence of grid adaptation on prediction of free surface at sluice feed.

A primary concern of the modelling is that the axial velocity profiles match the Law of the Wall near the sluice discharge. This is influenced by both the quality of the free surface prediction and also by the turbulence models.

Figure 3 shows the predicted axial velocity profiles at 1.4m from the feed end of the sluice for a feed flow rate of 0.917 kg.s^{-1}. At this axial position the flow is fully developed. The Standard K-omega model (STKW) gives the best agreement with the Law of the Wall, whilst the Shear Stress K-omega model (SSKW) over predicts the velocity somewhat at a $y^+ \sim 100$. Both the RKE and RSM models show a droop in the velocity profiles as the free surface is approached. It should also be noted that the RSM model significantly over predicts the height of the free surface (3.53 mm) when compared to the other models (eg 2.1mm for STKW).

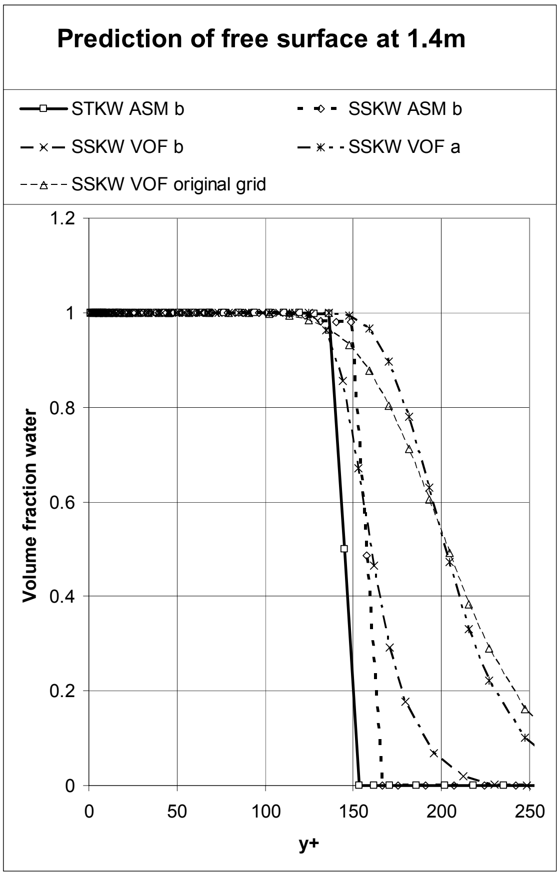

Figure 2 – Influence of grid adaptation and free surface models on prediction of the water free surface at 1.4m. Splitter height 2.2 mm

The poor agreement of the RKE and RSM models arises because these models used Fluent's enhanced wall functions. At the feed flow rate used in the simulations shown in Figure 3, the turbulent Reynolds number never exceeded 200 across the entire water flow. Hence the package assumed that the entire water flow was wall affected and did not use the dissipation as modelled with the ε transport equation, but rather calculated the length scale for the eddy viscosity algebraically using the one equation model of Wolfenstein (1969). This effectively eliminates any influence of the free surface on the modelled turbulent length scale and thus the turbulent viscosity is over predicted near the free surface with the RKE and RSM models, with a consequent droop in the predicted velocity.

The standard k-omega model is superior in that it can model the laminar to log transition region using the full two equation turbulence model, instead of the enhanced wall function approach used by the RKE model and RSM models. Thus in sluice simulations the full two equation turbulence model is used across the entire water flow domain. While the eddy viscosity is not modelled as well as it could be, it is apparent that free surface does introduce a peak in the specific dissipation and this causes the eddy viscosity to fall as the free surface is approached, such that the velocity profile gives a reasonable agreement with the Law of the Wall.

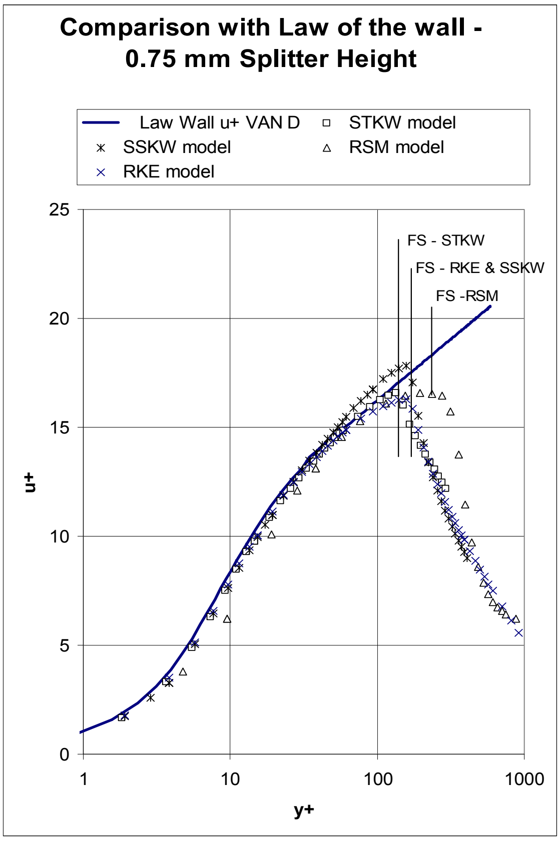

Figure 3 – Prediction of axial velocities compared with Law of the Wall at 1.4 m from the sluice feed as a function of turbulence model for a feed flow rate of 0.917 $kg.s^{-1}$.

Figure 4 shows the velocity profiles as a function of feed flow rate using the standard k-omega model. Clearly the model predicts a small droop in the velocity near the free surface and this is probably as a good a fit to the expected Law of the Wall behaviour that can be obtained with a basic two equation turbulence model.

Figure 5 shows a comparison between the flow depth predicted by the CFD at 1.4m from the feed and that measured experimentally. These results show that the CFD predicts the flow depth quite well.

CONCLUSION

CFD simulations of flows in gravity sluices have been conducted using Fluent. Best prediction of the free surface is obtained with grid adaptation at the feed end of the sluice and a sharper free surface position is obtained with the Algebraic Slip Mixture Model than is obtained with the Volume of Fluid Model.

The standard k-omega model gives the best agreement with the Law of the Wall in the fully developed region, although it predicts a slight droop in the water velocity near the free surface compared to the expected Law of the Wall behaviour. A better turbulence model is probably needed if a closer agreement with the Law of the Wall is necessary. Fluid heights predicted by the CFD agree well with those measured experimentally.

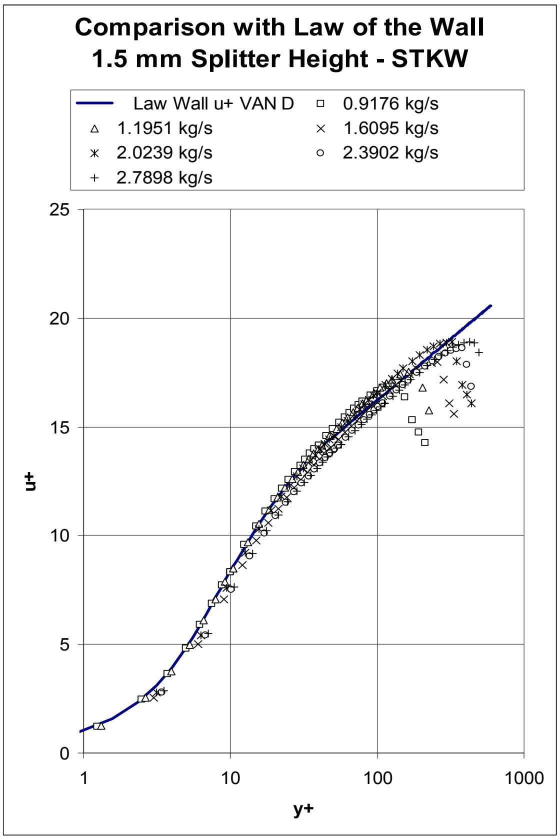

Figure 4 – Prediction of axial velocities compared with the Law of the wall at 1.4 m from the sluice feed as a function of feed flow rate using the standard k-omega model.

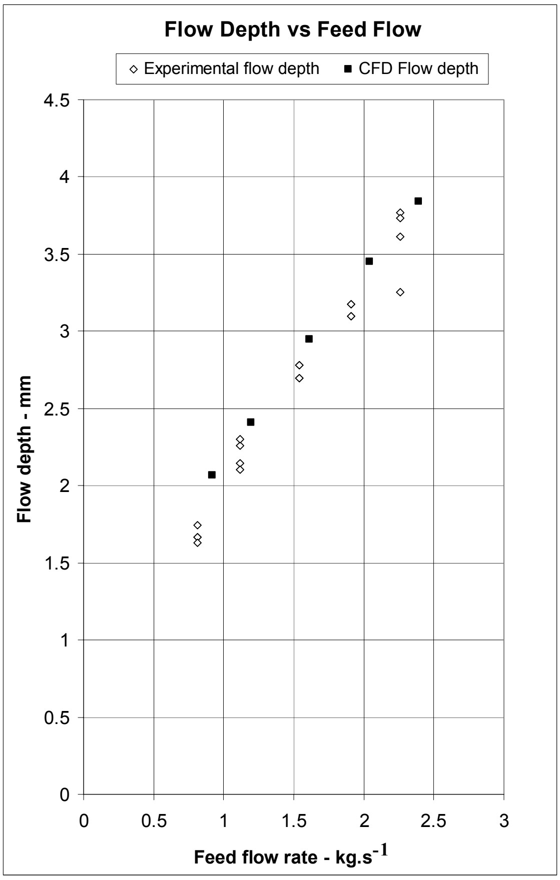

Figure 5 – Comparison between flow depth measured experimentally in sluice and that predicted by CFD models at 1.4 m. Standard k-omega model with ASM model.

REFERENCES

HIRT, C., W., and NICHOLS, B., D., (1981), "Volume of fluid (VOF) method for the dynamics of free boundaries", *J Comp. Phys.,* **39**, 201-225

HUNT, J. N., (1964), "Incompressible Fluid Dynamics", Longmans, London.

MAJUMDER A. K., (2002) "Modelling of Flowing Film Concentrators", Ph D Thesis, JKMRC, University of Queensland.

MANNINEN, M., and TAIVASSALO, V. (1996), *On the Mixture model for multiphase flow,* Valtion Teknillinen Tutkimuskeskus, Espoo, Finland.

NAKAYMA, A. and YOKOJIMA, S., (2001) "Direct numerical simulation of the fully developed open-channel flow at subcritical froude numbers", in LIU, C., SAKELL, L., and Beutner, T., (eds.) *DNS/LES Progress and Challeng*es, Greyden Press, Columbus, 569-576.

NEZU, I. and RODI W., (1986), "Open-channel flow measurements with a laser Doppler anemometer", *J. Hydraulic Eng,* **112,** 335-355.

WILCOX, D., C., (1988), "Reassessment of the scale determining equation for advanced turbulence models". *AIAA J.,* **26,** 1299-1310.

WILCOX, D. C., (1998), *Turbulence modeling for CFD*, DCW Industries, La Canada, California.

PRINOS, P., and ZERIS, A., (1996), "Uniform flow in open channels with steep slopes", *J Hydraulic Res.,* **33,** 705-718.

SHIH T.-H., LIOU, W., W., SHABBIR, A., and ZHU, J., (1995) "A New k-ε eddy-viscosity model for high reynolds number turbulent flows - model development and validation", *Computers Fluids*, **24**, 227-238.

WOLFSTEIN M. (1969),"The Velocity and Temperature Distribution of One-Dimensional Flow with Turbulence Augmentation and Pressure Gradient.", *Int. J. Heat Mass Transfer*, 12,301-318