# Computational and Experimental Investigation of Spiral Concentrator Flows

**B W Matthews[1,2], P Holtham[3], C A J Fletcher[1], K Golab[3] and A C Partridge[2]**

## ABSTRACT

Spiral concentrators are used globally in the fine coal processing industry to segregate particles, by gravitation, on the basis of density and size. Consisting of an open trough that twists vertically downwards about a central axis, a slurry mix of particles and water is fed to the top of the concentrator. Particles are then separated radially as they gravitate downward. Since their introduction to Australia in the 1940's, the generic design has evolved largely by laboratory trial-and-error investigations of different prototypes. However, this approach has proven expensive and optimal designs have not been necessarily developed. Accordingly, Computational Fluid Dynamics (CFD) analysis has been used recently as an alternative method of investigation, to assume as is envisaged, a role in the design process. To date, CFD models have progressed to simulations of turbulent fluid flow on current production spiral designs, and are continuing to be adapted for inclusion of particles at realistic feed concentrations. To be able to use the models confidently however, laboratory experiments must also be performed to validate the predictions during the development stage. This paper reports the current findings of an ongoing CFD and experimental program applied to one spiral unit. Satisfactory quantitative agreement has been achieved for the fluid and particulate flow characteristics, and although further validations are appropriate, the model already possesses significant potential for use as a reliable predictive design tool.

## INTRODUCTION

Spiral concentrators consist of an open trough that spirals vertically downwards in a helix configuration. Fine 100-1500 μm coal and waste rock particles are fed at concentrations of 30-40% by volume, and are segregated radially on the basis of density and size as they gravitate downward. Ideally, light suspended particles (coal) travel to the outer trough regions whilst heavy particles (quartz) settle and move inwards toward the central column. To optimise this process, historical evolution of the design has been almost exclusively based on empirical trial-and-error developments of the appropriate geometry. However, this approach has proven to be expensive and is hence somewhat prohibitive.

Recognising that a fundamental understanding of the flow physics could lead to the development of a predictive model in the design process, Holtham (1990) investigated by experimentation the fluid flow patterns and particle movements on the LD9 spiral unit. This study has since formed the basis of an improved and continuing experimental program (Golab *et al.*, 1997). In parallel with these programs, modellers this decade have turned from using predominantly analytical approaches (Holland-Batt, 1989) to Computational Fluid Dynamics (CFD) analysis which requires much less empirical input. Following the preliminary investigations of laminar fluid flow on the LD9 unit (Jancar *et al.*, 1995; Matthews *et al.*, 1996), the models have been extended to include both turbulence within the fluid phase and dilute particulate additions (Matthews *et al.*, 1997; 1998).

Attempts to understand the mechanisms of spiral concentrator operation have been restricted by the complex character of the flow. Even when considering the fluid phase without inclusion of particles, the flow possesses a free surface, is thin with depths < 1 cm typically, and displays laminar to increasingly turbulent behaviour radially outwards with maximum velocities reaching 3 m/s. Moreover, a secondary circulation current in a plane perpendicular to the mainstream flow direction travels outwards near the free surface and back inwards toward the central column near the trough base. The complexity is further magnified by the mutual interaction of the fluid flow and particles, by interactions between the particles themselves, and by the obvious constraints imposed on the flow by the geometry.

[1] CANCES, University of NSW
[2] School of Mining Engineering , University of NSW
[3] Julius Kruttschnitt Mineral Research Centre, University of Queensland

In this paper, the current experimental procedures and computational methods of investigation, used to quantify the relevant properties of the flow, are briefly described. To assess the reliability of the predictive model, validations for the LD9 unit at industrial flow rates are presented for the fluid flow. Numerical results for particulate additions at dilute concentration (to examine the influences of the hydrodynamic processes) and concentrations where particle-particle interactions become significant are then given. The practical significance and potential worth of the model are finally discussed.

## INVESTIGATIVE PROCEDURES

### Experimentation

Experimental investigations have been performed on the LD9 unit at industrial flow rates of 4, 6, and 8 $m^3$/hr. Holtham (1990) measured flow depths using a computer controlled multi-point depth gauge equipped with conductivity probes. Flow rates were used to estimate mainstream velocity components for eight radial sections across the trough, and by the injection of dye tracers, the secondary flow was identified and measured. Golab *et al.* (1997) have measured instantaneous fluid velocities on the LD9 unit using Particle Image Velocimetry (PIV). In the PIV procedure, rhodamine particles are injected into the flow and illuminated with a laser beam, pulsed on-and-off at 1 ms intervals. Particle movements are recorded by videotape, the images digitised, and the velocities and positions of the tracers calculated by computer. In order to introduce the laser sheet to the flow, a 30° section of the original fibreglass trough is removed and replaced with transparent acrylic (Fig. 1). Due to the curvature of the trough, five separate windows are required through which the laser beam is transmitted and, by adjusting the position of the laser within each window, velocity measurements can be made over most of the trough region.

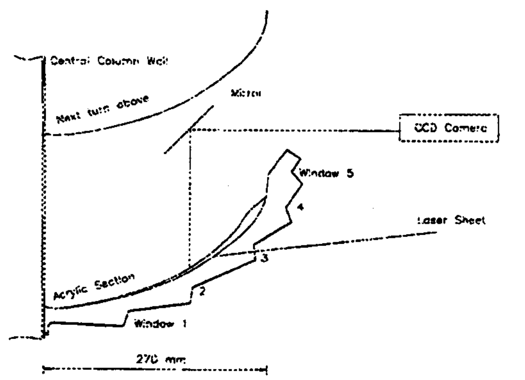


**Fig. 1 - Design of the acrylic section and general arrangement of the PIV apparatus**

To investigate the effects of the hydrodynamic processes on particulate transport, Holtham (1990) measured first the discharge positions of quartz particles fed at dilute (0.3% by volume) concentration and fluid flow rate of 6 $m^3$/hr. Three narrow size distributions with mean diameters of 75, 530 and 1400 μm were considered, and the percent recovery to each of 8 radial streams was calculated. Experiments using quartz with a broad (100-1000μm) grain size distribution were also conducted at a volume concentration of 15%. This was done primarily to assess the significance of the Bagnold forces, as high volume fractions of particles (up to 50%) were found to accumulate in the inner region. Currently, the PIV technique for the fluid flow is being adapted to examine particles with material densities of that of coal and quartz.

### Computation

The CFD model solves the steady-state Reynolds-averaged Navier-Stokes equations using the commercial software FLUENT. The Volume of Fluid (VOF) and RNG k-ε turbulence formulations are used to simulate the free surface transport and laminar to turbulence transition across the trough (Matthews *et al.*, 1996, 1997). The LD9 computational domain is given in Fig. 2. At the inlet, the volume fraction of the water phase is arbitrarily "guessed" and the velocities specified to give a desired flow rate. The domain is divided into 35° sections with the outlet solution specified as the inlet conditions for the next downstream sector until fully developed flow conditions are obtained. Four walls bound the domain and the flow is essentially a duct flow that includes an interface between water and air. The mesh contains respectively, 20 x 39-46 x 208 control volumes in the mainstream, depth-wise and radial directions, with cells clustered toward the spiral base.

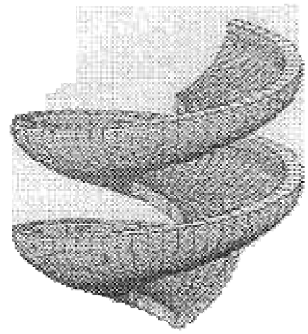

**Fig. 2 - LD9 computational domain with reduced cells for clarity**

There are two main methods used to model particle flows using CFD. In the *Lagrangian* method, particles are treated as discrete entities and their trajectories calculated as they proceed through the fluid flow field. This approach yields detailed descriptions of the flow of individual particles, and is a more fundamental procedure to describe the acting hydrodynamic forces (pressure gradient, buoyancy, drag and virtual mass) and collisions with a wall. Alternatively, the *Eulerian* approach regards the particle cloud as another continuum able to interact and inter-penetrate with the fluid flow. Suited to particle flows at high concentration, interactions between particles are accounted for and mean particulate velocities and volume concentrations are calculated throughout the flow domain.

Matthews *et al.*, (1998) have examined particle flows at dilute feeds using the Lagrangian method. Although these processes are believed to be the main contributors to particulate separation, two-way coupling effects of fluid-particle interactions and the interplay between particles have yet to be investigated. These additional factors have now been examined using the Eulerian approach, and the results of a preliminary investigation are presented in this paper. At the present time, location of the free surface must be assumed in the analysis although an approximate profile can be employed from the fluid flow solution. By making an adjustment for the high accumulation of solids that typically arise in the inner region, this limitation is not too severe as measured profiles are reasonably similar, particularly at small feed concentrations (Holtham, 1990). A fully predictive Eulerian-VOF free surface model in FLUENT will be available by mid 1998.

## EXPERIMENTAL VALIDATION OF MODEL USING THE LD9 UNIT

CFD simulations of the fluid flow have been published previously (Matthews *et al.*, 1997; 1998) and are found to be in satisfactory agreement with the experimental data (Holtham, 1990; Golab *et al.*, 1997). In Fig. 3, the fully developed transverse profile with predicted and measured depth comparisons normal to the trough are given. The simulated mainstream velocity distribution is also depicted in the top part of Fig. 3. The discrepancy in the outermost region reflects clearly detectable air entrainment, currently unaccounted for by the model, which has the effect of "bulking" the flow. Caused by turbulent eddies escaping above the free surface and entrapping air as they return as droplets to the medium, the entrainment is likely to increase the water depth by 15-20% for the characteristic Froude numbers encountered in the outer zone (Matthews *et al.*, 1998).

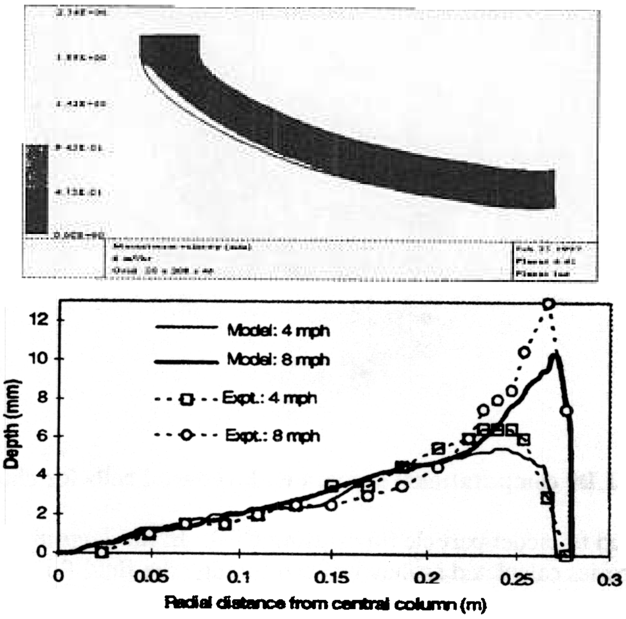


**Fig. 3 - Predicted fluid profile at $6 m^3/hr$ (top) and depth comparisons with experimental data at 4 and 8 $m^3/hr$ (bottom)**

Perhaps the most encouraging aspect of the fluid flow simulations has been the satisfactory comparisons of mean mainstream velocity with the PIV measurements at arbitrary depths and radii within the flow. Predicted and measured mainstream velocities at 6 $m^3/hr$ and 1 and 5 mm depths are given in Fig. 4. Also given in Fig. 4 is the classic structure of the secondary flow, first identified and measured using dye injections (Holtham, 1992). The PIV measurements and computer simulations reveal that the secondary flow is transient in nature with velocity fluctuations of at least the same order as the mean values. Characteristic values for the mean secondary motion are generally an order of magnitude less than the mainstream velocities and vary from 0.01-0.3 m/s.

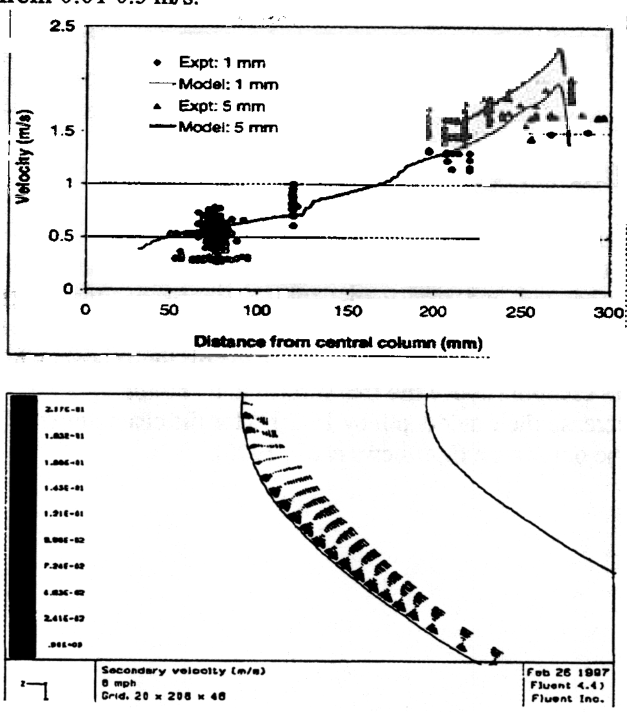


**Fig. 4 -Top: comparison of simulated and measured mainstream velocities at depths of 1 and 5 mm at 6 $m^3/hr$; bottom: Secondary flow in the outer trough at 8 $m^3/hr$**

The development of a fully predictive, robust and accurate model for the fluid flow has been critical. Indeed, the evidence suggests that the detailed flow character primarily drives the separation processes (Holland-Batt, 1989). Having established a reliable model for the fluid phase, the next step is to examine the particulate flow at dilute concentration, based on the acting hydrodynamic forces using the Lagrangian method. Although experimental validation is not yet available, the nature of a particle-wall collision is assumed to obey the impulse equations describing sliding upon impact (Sommerfield, 1992). Sliding collisions are believed to predominate over non-sliding contacts as typically impact angles are small, and particle diameters much larger than the laminar sub-layer thickness (Matthews *et al.*, 1998).

The Lagrangian procedure consists of injecting 100 particles of the same density and size uniformly across the trough on the free surface of the flow. The trajectories are then calculated until the radial positions of hydrodynamic equilibrium are reached, typically after 3-4 turns of the helix. Analyses were performed for 100-1500 μm coal and quartz particles with material densities of 1450 and 2650 kg/m$^3$, respectively. The results suggest that the LD9 concentrator is able by hydrodynamic processes to separate particles in the somewhat narrow size range of 200-500 μm. Within this range, the classic pattern is observed with coal particles migrating to the outer regions whilst quartz accumulates at the small radii (Fig. 5). At < 100 μm, both coal and quartz are predicted to remain in suspension and move to the outermost zone; above 500 μm both particle types are found to accumulate within the inner regions.

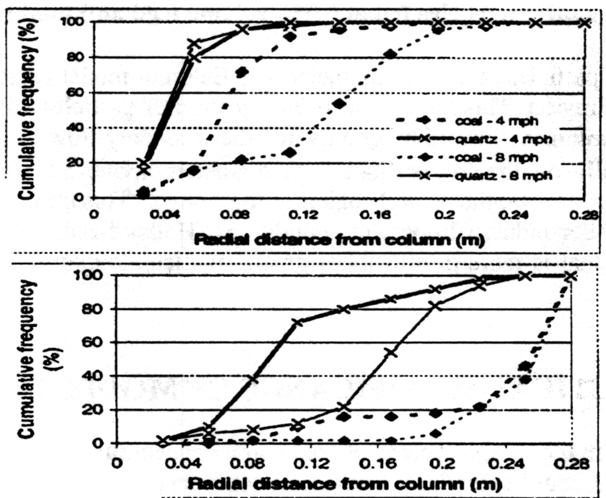


**Fig. 5 - Radial distributions of 0.5 mm (top) and 0.2 mm (bottom) diameter coal and quartz particles at fluid flow rates of 4 and 8 m$^3$/hr (mph): Using the Lagrangian analysis**

At the higher flow rate, the upper extent of particulate separation is extended from 200 to 500 μm as magnitudes of the secondary velocity are significantly stronger; at 4 and 8 m$^3$/hr, the predicted maximums of $u$ are 0.13 and 0.22 m/s, respectively. However based on the hydrodynamic processes alone, no apparent single definitive radial cut exists between the desired product and waste material (Fig. 5). For example at 8 m$^3$/hr, optimum separation of 200 μm particles would be achieved at ~0.20 m from the central column. Conversely, poor separation at 0.20 m would occur for particles of 500 μm diameter, for which segregation should be performed at ~0.06 m radius according to the analysis. Hence, if it is accepted that coal concentrators are designed so that the hydrodynamic processes are the dominant factors controlling separation, then the simulations suggest that the LD9 performance could be improved.

Differences have been found when the Lagrangian radial distributions for quartz are compared with the measurements obtained by Holtham (1990) at dilute 0.3% volume feed. First, Holtham discovered that approximately half of the smallest particles of < 100 μm diameter accumulated within the inner region, although the rest migrated to the outer radii as predicted by the model. Second, particles with progressively larger size above 500 μm migrated outwards to accumulate predominantly within the central portion of the trough. Accordingly, the Eulerian method was employed to simulate

Holtham's experiments of 75, 530, and 1400 μm quartz feeds at 0.3% volume concentration and fluid flow rate of 6 $m^3$/hr. Significantly improved comparisons with the measured radial distributions of particles have been found, and are depicted in Fig. 6.

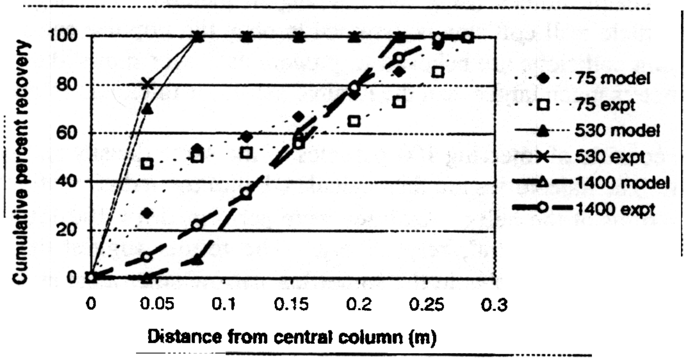


**Fig. 6 - Comparisons of measured and predicted Eulerian radial distributions of 75, 530 and 1400 μm quartz particles fed at 0.3% volume feed and fluid flow rate of 6 $m^3$/hr**

Considering first the finest particles of 75 μm diameter, the Eulerian model correctly predicts that a large proportion of particles move to the inner region. This reflects an even injection of particles with depth whereby those near the trough base in the low velocity region move inward by gravity and the secondary flow. Conversely, all particles in the Lagrangian analysis were placed initially on the free surface. At diameters above 500 μm the particles are predicted to move progressively further outward toward the central region of the trough. This trend has been measured in the experiments of Holtham (Fig. 6), occurs under industrial operating conditions (Holland-Batt, 1989), and can be attributable primarily to Bagnold's dispersive force that is more prominant at greater particulate diameters and volume concentrations (Holtham, 1992).

## PRACTICAL SIGNIFICANCE OF MODEL DEVELOPMENT

Currently, CFD is not used in the design process of coal spiral concentrators. The accurate and robust nature of the model however, suggests that CFD could potentially assume, at some level, a significant role. For example, the detailed fluid flow solutions of transverse profiles and velocity distributions (without even considering particle inputs) may be of use to an experienced design engineer. The ability to extract detailed information above that available through experimentation, can also lead to further fundamental insights into the behaviour of the flow. Perhaps more importantly, because the hydrodynamic processes are believed to be the dominant factors driving separation, the Lagrangian analyses can be used as a preliminary measure in assessing the potential performance of any given prototype. Such information would probably be useful without considering the effects of particle-particle interactions, which the Eulerian analysis suggests, should be considered in the complete analysis.

Fundamental information of the fluid flow properties and particle trajectories can be attained much faster computationally than experimentally. To obtain a fluid flow solution for an arbitrary geometry requires 48-72 hours using a single processor of a 120 MHz UNIX server. A further 12 hours are required to perform the dilute particulate flows for a range of particle types using the Lagrangian method. In contrast, many weeks would be needed to achieve experimentally the same detailed information for the fluid flow using the procedures outlined in this paper. Moreover, the computer hardware employed in the investigation is by no means state-of-the-art, so that continuing rapid technological developments are likely to see costs of available computers decline significantly. Indeed, comparative fluid flow solutions presented 18 months ago (Matthews *et al.*, 1996) required two weeks of processing on a HP 7000 series work station. It is then feasible that projected solutions in the near future will be affordably accomplished within several hours.

Although the fluid flow predictions are generally excellent and the particulate simulations encouraging, further improvements to the model and experimental validation are necessary. The most notable numerical addition will be combination of the VOF and Eulerian models to fully simulate the free surface particulate flows at realistic feed concentrations. Experimental validation will focus upon the particulate flow, namely the measurement of particle velocities, dynamics of particle-wall collisions (including restitution and friction coefficients) and modes of particle transport. By continuing to implement realistic physical aspects of the flow into the model, and as the costs of hardware continue to decline, CFD will gain greater potential for use in the design process. Indeed, there may already be economic benefits in using the current state of model development, reported in this paper, as a design tool at some level. Ultimately, it is envisaged that the CFD model will eliminate most of the competing designs during the prototype development stage, to perform physical testing on one or two preferred configurations for the given application.

## CONCLUSIONS

This paper has reported the current findings of a continuing experimental and computational study of spiral concentrator flows applied to the LD9 unit. The aim of this study is to develop and validate a CFD model that can be used in the design process. Accurate results for the fluid flow characteristics have been simulated and the model is robust enough to be used for arbitrary geometric configurations. Particulate analyses have been performed at both dilute and high concentrations to examine the influences of the hydrodynamic processes and particle-particle interactions. Although further extensions to the model and accompanying experimental validation are required, particularly with respect to the particulate flow at realistic feed rates, predictions of the model are encouraging. Indeed, the current state of model development is likely to be of some use at the design level, and this will become increasingly the case as future costs of computer hardware decline and improvements to the model are implemented.

## REFERENCES

Golab, K J, Holtham, P N and Wu, J, 1997. Validation of a computer model of fluid flow on the spiral separator, in *Proceedings Innovation in physical separation technologies, Richard Mozely Memorial Symposium* (Institution of Mining and Metallurgy: London).

Holland-Batt, A B, 1989. Spiral separation: theory and simulation. *Trans. Instn. Min. Metall. (Sect. C)*,. 98: C46-60.

Holtham, P N, 1990. The fluid flow pattern and particle motion on spiral separators. Unpublished Ph.D. Thesis, University of New South Wales.

Holtham, P N, 1992. Particle transport in gravity concentrators and the Bagnold effect. *Minerals Engineering*, 5(2): 205-221.

Jancar, T, Fletcher, C A J, Holtham, P N and Reizes, J A, 1995. Computational and experimental investigation of spiral separator hydrodynamics, in *Proceedings XIX International Mineral Processing Congress*, (San Francisco).

Matthews, B W, Fletcher, C A J, Partridge, A C and Jancar, T, 1996. Computational simulation of spiral concentrator flows in the mineral processing industry, in *Proceedings CHEMECA '96* (Australasian Institution of Chemical Engineering: Sydney).

Matthews, B W, Fletcher, C A J, Partridge, A C, 1997. Computational simulation of fluid and dilute particulate flows on spiral concentrators, in *Proceedings International Conference Computational Fluid Dynamics in Mineral & Metal Processing and Power Generation* (CSIRO Division of Minerals: Melbourne).

Matthews, B W, Fletcher, C A J, Partridge, A C, 1998. Computational simulation of fluid and dilute particulate flows on spiral concentrators. *Applied Mathematical Modelling.*

Sommerfield, M, 1992. Modelling of particle-wall collisions in confined gas-particle flows. *Int. J. Multiphase Flow*, 18(6): 905-926.