Computational Modelling of Coal Slurry Distributor Flow

D. Yang[1,2], C. A. J. Fletcher[1] and A. C. Partridge[2]

1. CANCES, The University of New South Wales, Sydney 2052
2. Center for Mineral Engineering, The University of New South Wales, Sydney 2052

ABSTRACT

Sputnik distributors are widely employed by the coal industry as the first stage of coal processing in the plant. Coal and water are mixed to produce, ideally, an even distribution among the outlets. However it is generally recognised that there are often serious problems associated with very poor performance - especially that of uneven distribution. Experimental investigations to date have been concerned with the input/output relationship, but reveal little about the nature of the internal mixing processes. There is a necessity to examine and quantify the distributor's internal two-phase fluid dynamic processes using a numerical model. The RNG k-ε model is employed to simulate high speed swirling flow which occurs within the distributor. The Lagrangian method is used to simulate the particle motion and provides a qualitative prediction for the coal slurry behavior.

1. INTRODUCTION

Sputnik distributors are widely employed in coal preparation to mix coal, fed from the top of the distributor and water injected by a number of tangential inlets in the upper chamber of the distributor, and split the slurry into a number of outlet streams downstream. Due to the complexity of the particulate behaviour, coal separation processes of all types have one common characteristic: they are imperfect (Partridge, 1992)! The Sputnik distributor shares this problem too. It is generally recognized that there are often serious economic problems associated with very poor performance, especially that of uneven distribution. Pitt and Stone (1989) estimated that the cost of lost yield from dense medium cyclones due to poor distribution, was between $0.5-1.0 million per annum for large plants. This problem has been experienced for 15 years in Australia (Holtham, 1995). Some experimental investigations of distributors, to date, have been concerned only with its input/output relationships, and the results are still not satisfactory because, unfortunately, little is known about the internal processes. This highlights the necessity of using a mathematical model to examine and quantify the distributor's detailed internal fluid dynamics.

Due to the complex geometry of the Sputnik distributor and its internal high speed swirling flow, the distributor performance demonstrates the unpredictability of the present distribution process, in that no simple physical model or theoretical analysis so far has provided satisfactory solutions. Naturally, one will think of using the techniques of Computational Fluid Dynamics (CFD) to investigate the coal slurry flow, as the effectiveness of CFD has increased rapidly with the significant increase in computer power during the past two decades. In the aerodynamic, mechanical, and hydraulic design areas etc., CFD has matured to the point where it is widely accepted as a key design tool since it provides advantages over experimental testing by providing detailed, comprehensive, cost-effective and time efficient information. However in the mineral processing industry, the application of CFD has only a few years history. Good reviews of recent developments of mathematical models in mineral processing have been given by Schwarz (1991) and Fletcher et al (1995).

2. THEORETICAL CONSIDERATIONS

Generally, there are two prediction approaches for the two-phase flows, namely Lagrangian and Eulerian methods. The Lagrangian method predicts the individual particle trajectory and velocity in the fluid phase as a result of forces acting on the particle, but cannot give easily information about the mean particulate velocity and concentration. Treating the particles as a continuum, and solving the appropriate continuum equations for the fluid and particles, is referred to as the Eulerian approach. At the present, it is not universally accepted which approach is more valid for simulating the multiphase turbulent flow. Numerically, the Eulerian approach will be more straightforward when the particles input stream is continuously fed with a high concentration, but the Lagrangian approach will give particle trajectory information, in a way provides more direct visual impact. However with the particle phase concentration becoming high, coupling of the particulate phase and its carrier fluid at both the mean flow and turbulent levels is becoming more important and very complicated. At present completely solving the multiphase flow is out of the question because of limited understanding of the fluid-particle and particle-particle interactions. But many researchers from different fields have been studying this topic using both the experiments and the mathematical models. For the particle phase alone, the particle-particle collision stress and particle stress are reasonably modelled using the kinetic theory (Jenkins and Savage (1983), Johnson and Jackson (1987), Ding and Gidaspow (1990)). But questions of the interactions between the two phases at the fluctuation level still remain. For the carrier fluid phase, some common characteristics of the turbulent properties have been observed through a number of the measurements for both the dilute and dense flows (Tsuji et. al, 1984, Zisselmar and Molerus, 1979). It is observed that the presence of the particulate phase will have a significant effect on the turbulence structure of its carrier phase, and generally will cause reductions of the turbulent energy and dissipation, but under a special condition, for instance an appropriate particle size, the particulate phase will enhance the intensity of turbulence. The reason is that big particles will cause separation of the surrounding fluid resulting in wakes behind the particles. Numerically, Elghobashi and Abou-Arab (1983), Chen and Wood (1985), Tu and Fletcher (1995) have considered such effects in their respective turbulent models.

At the present stage of this work, to obtain a visual interpretation of the particle motion, the continuum theory is applied to the air and water phases and the particle phase is treated by a dispersed approach-the Lagrangian method.

2.1 Governing Equations of Air-Water Phase Flow

In practice, the flow within the distributor is a three-phase (solid-air-water) flow. The particles firstly contact the air phase in the upper chamber of the distributor, and then are mixed with the water phase after passing an interface between the air phase and water phase.

As air and water are immiscible, there must be an interface between them. A single set of momentum equations will be shared by both the air phase and the water phase through a volume fraction of the fluid. The volume fraction of each fluid is determined throughout the computational domain by a mass balance law. Mathematically, after Reynolds averaging the governing equations can be expressed as:

continuity equation for the *kth* phase:

$$\frac{\partial \varepsilon_k}{\partial t} + u_i \frac{\partial \varepsilon_k}{\partial x_i} = 0 \tag{1}$$

where ε_k is the volume fraction of the *kth* phase, u_i is the velocity component in the i direction of Cartesian coordinates;

momentum equation:

$$\frac{\partial}{\partial t}(\rho u_j) + \frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}(\mu_t[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}] - (\frac{2}{3}\mu_t \frac{\partial u_l}{\partial x_l})) + \rho g_i + \frac{\partial}{\partial x_j}(\overline{\rho u'_i u'_j}) \tag{2}$$

in which p is the static pressure; g_i and is the gravitational acceleration in the i direction, $-\overline{\rho u'_i u'_j}$ is the Reynolds stress; μ_t is the effective viscosity, defined as the sum of turbulent viscosity and molecular viscosity, μ, and ρ is the density for the fluid which is determined by the presence of the component phases in each control volume. For the N phase system, the volume fraction averaged density and viscosity take the forms:

$$\rho = \sum \varepsilon_k \rho_k \qquad (3)$$

$$\mu = \sum \varepsilon_k \mu_k \qquad (4)$$

For the water-air two phase flow, the volume fractions should follow

$$\varepsilon_w + \varepsilon_a = 1 \qquad (5)$$

where ε_w denotes the volume fraction of the water, and ε_a stands for the volume fraction of the air. But in most of the region the volume fraction of the water and the air is either equal to 1 or 0.

As high speed swirling flow dominates within the distributor, the flow is inevitably turbulent. Predicting turbulent flow requires appropriate modelling procedures to describe the effect of turbulent fluctuations of velocity on the basic mass conservation and momentum equations. Hence, a turbulence model will be required to close the system of equations. The main task of the turbulence model is to produce expressions or closure models that allow the evaluation of these correlations in terms of mean flow quantities. The turbulence closure model used in this study is the two equation RNG k-ε model (Yakhot and Orszag, 1986). The transport equations for the turbulent energy, k, and its dissipation rate, ε, are:

$$\frac{\partial k}{\partial t} + u_i \frac{\partial k}{\partial x_i} = \nu_t s^2 - \varepsilon + \frac{\partial}{\partial x_i}(\alpha \nu_t \frac{\partial k}{\partial x_i}) \qquad (6)$$

and

$$\frac{\partial \varepsilon}{\partial t} + u_i \frac{\partial \varepsilon}{\partial x_i} = c_{1\varepsilon} \frac{\varepsilon}{k} \nu_t s^2 - c_{2\varepsilon} \frac{\varepsilon^2}{k} - R + \frac{\partial}{\partial x_i}(\alpha \nu_t \frac{\partial \varepsilon}{\partial x_i}) \qquad (7)$$

where $\nu_t = c_\mu \frac{k^2}{\varepsilon}$, and the rate of strain term R is given by

$$R = \frac{c_\mu \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3} \frac{\varepsilon^2}{k} \qquad (8)$$

where $\eta = s k/\varepsilon$, $\eta_0 \approx 4.38$, and $s^2 = s_{ij} s_{ij}$ is the modulus of the rate strain tensor. Constants are given empirically as:
$c_{1\varepsilon} = 1.42, c_{2\varepsilon} = 1.68, \alpha = 1.39, c_\mu = 0.09$.

2.2 Lagrangian Formulation

Before discussing details of the Lagrangian approach, an important parameter, namely the Stokes number, S_t, is introduced first. The Stokes number determines the degree of interaction between the particle and its carrier fluid, and is defined as the ratio of the response time due to aerodynamic drag force on a particle to a turbulent eddy characteristic time of the fluid. Mathematically, it has a form

$$S_t = \rho_p D_p^2 / (18\mu) / t_e \qquad (9)$$

where D_p is the particle diameter, ρ_p is the density of the particulate phase and μ is the viscosity of carrier flow. The turbulent eddy characteristic time of the carrier, t_e, is given by

$$t_e = 0.2k / \varepsilon \qquad (10)$$

The Stokes number determines how an individual particle will respond to the fluid flow. When the Stokes number is large, say S_t>> 0.01, significant slip will occur between the particles and its carrier, i.e. the particle trajectories will differ substantially from the flow streamlines. As long as the Stokes number is small enough, S_t<<0.01, the particles will follow the fluid motion. If the concentration of the particulate phase is very small, then this kind of flow behaves as a single-phase flow. This aspect also can be explained from the turbulent point of view, if the particle is large compared to the scale of turbulence, the main effect of the turbulence will be on the drag coefficient and the particle will follow the slower large-scale turbulent motion of the fluid. If the particle is small compared to the smallest scale of the turbulence, it will respond to all the turbulence components of the fluid. Therefore the turbulent effect on the particle trajectories has to be considered. A good approach is given by Zhou & Leschziner (1991) and named the Discrete Random Walk (DRW) model. However, such model is only suitable for the small Stokes number and small size particles. Since the common particle diameter within the distributor is about 1 mm and the particle density has a value 1250 kg/m^3, the particle Stokes number will be large. The turbulent effects on the particle phase therefore, will be neglected in the Lagrangian approach.

For a given fluid, the equation describing the force balance with the inertia forces acting on the particle which is firstly given by Basset, Boussinesq, and Oseen can be written (for the x direction in Cartesian coordinates) as:

$$\frac{du_p}{dt} = F_d(u - u_p) + g_x(\rho_p - \rho)/\rho_p + F_x \qquad (11)$$

where $F_d(u-u_p)$ is the drag force per unit particle mass and

$$F_d = \frac{18\mu C_d \, \mathrm{Re}}{\rho_d D_p^2 24} \qquad (12)$$

here, u is the fluid phase velocity, u_p is the particle velocity, ρ is the density of the fluid, and R_e is the relative Reynolds number or the particle Reynolds number, which is defined as:

$$R_e = \frac{\rho D_p |u_p - u|}{\mu} \qquad (13)$$

The drag coefficient, Cd, is a function of the relative Reynolds number, and is of the following general form

$$C_d = a1 + a2/R_e + a3/R_e{}^2 \qquad (14)$$

where the a's are constants that apply over several ranges of R_e, as given by Morsi & Alexander (1972). The additional forces, Fx, include the 'virtual mass' force, the centrifugal and Coriolis forces, but the Basset term which constitutes an instantaneous flow resistance is neglected because the steady flow is considered. Finally, the additional forces, Fx, have a form

$$F_x = \frac{1}{2}\frac{\rho}{\rho_p}\frac{d}{dt}(u - u_p) + (1 - \frac{\rho}{\rho_p})\omega^2 x + 2\omega(u_{y,p} - \frac{\rho}{\rho_p}u_y) \qquad (15)$$

in which ω is the particle angular velocity.

Integration of Eq. (11) in time yields the velocity of the particle at each point along the trajectory, with the trajectory itself predicted via:

$$\frac{dx}{dt} = u_p \qquad (16)$$

During the integration, the fluid phase velocity, u, can be taken as the cell-based velocity for all positions within the cell. Assuming that the term containing the body force remains constant over each small time interval, and linearizing any other forces acting on the particle, the trajectory equation can be rewritten in a simplified form as:

$$\frac{du_p}{dt} = (u - u_p)\alpha \qquad (17)$$

which can be integrated to yield:

$$u_p(t + \Delta t) = u_p(t) + (u - u_p(t))(1 - e^{-\alpha(t+\Delta t)}) \qquad (18)$$

and α is a coefficient and should be set in the range 0.1-0.3.

3. NUMERICAL SCHEME

A commercial code Fluent (V4.4) is employed in this project for saving the time of redeveloping the code. Fluent uses a control volume based technique to solve the conservation equations for mass, momentum, energy, and turbulent quantities. This approach consists of: (1) Division of the domain into discrete control volumes using a general curvilinear grid; (2) Integration of the governing equations on the individual control volumes to construct the algebraic equations for discrete unknowns (velocities, pressure, etc.); (3) Solution of the discretised equations. Comprehensive discussion of the finite volume method and the differencing schemes are given by Patankar (1980) and Fletcher (1991).

4. COMPUTATIONAL RESULTS AND DISCUSSIONS

In parallel with the numerical studies of the distributor flow, an experimental investigation is operational at Julius Kruttschnitt Mineral Research Center (JKMRC) of the University of Queensland (Kelly and Holtham, 1996). Due to the timing, at the present the experimental data for the both phases are not ready to compare with results produced by this work, but it is expected that in the future the flow data within the distributor will be collected to compare with and calibrate the mathematical model developed by the present studies. Detailed dimensions used by this model can be seen in their work.

To reduce the computational cost and permit a more efficient control volume, the computational mesh has been generated by the Geomesh code (v3.1) based on a half physical model in order to take advantage of the symmetry flow condition. The main characteristics of the physical model, such as the tangential water inlets, the orifice and the distributor insert (bottom baffle), have been modelled (Fig. 1).

To investigate the biased coal slurry distribution for the Sputnik distributors, the first thing is to investigate the water phase flow without the solid phase because we maintain that the basic flow mechanism is the water 'pulls' the particles around the distributor and, finally, they are mixed together to flow out. The water inlet velocity has a value of 1.6 m/s. The water viscosity and density used in the model are 9.0E-4 kg/m-s, 1000 kg/m^3, while the air viscosity and density are 1.7E-5 kg/m-s, 1.29 kg/m^3, respectively.

At the beginning of the computation, an initial flat water surface above the inlets is assumed so that a converged solution can be more easily obtained. After running the case file for a certain period, if the water surface no longer changes with physical time, then a steady water surface and solution are obtained. Fig. 2 shows a typical high curvature interface between the water and the air. The reason is that a vortex motion is induced by a centrifugal force due to the swirling velocities. Fig. 3 shows the velocity distributions for both phases. It is quite interesting to see that in the upper chamber there is a vertical circulating region for the air, the air comes in from the core region at the top and comes out the distributor nearby the wall to maintain the mass balance. But the magnitude of the air velocities is much smaller than the water's. The maximum velocity of the water phase occurs at the outlets with a value of 2.4 m/s. Strong swirling velocities are generated in the upper chamber for the water phase, while relatively weak swirling velocities are found in the lower chamber. It is also observed that the swirling velocity is proportional to the local radius of the distributor in the upper chamber. But in the lower chamber due to the weak swirling motion of the water phase, the mixing process is less effective. Two extreme cases should be noted here. One case is when the inlet velocities or mass flow rates are very low, there will not be a fixed water surface in the distributor. Like a water fall in the distributor the water will be more directly drained out through the outlets. Another case is that too much water flows into the distributor beyond the capacity of the distributor resulting in a distributor overflow. Obviously these two cases would not be expected in the operational plant.

At two different locations of the top of the distributor, the injected particles (ρ=1250 kg/cm^3, D_p= 1 mm) have been tracked. One case is the particles released from the center, and another one is at half the radius of the distributor. It is observed that for the both the cases the particles in the upper chamber will pass straight through the air phase and start to be dragged immediately by the water phase due to the higher density of the water (Figs. 4 and 5), but the off-center feeding will generate a longer particle residence time. The reason is that in the upper chamber, the swirling velocity of flow is proportional to the radius of the distributor, and the particles are most likely driven by the high speed swirling flow. In other words the hydrodynamic force effect is significant nearby the wall region. By contrast, in the core region of the distributor, the particle is more likely subject to the gravity resulting in a shorter residence time. From this point of view, it might be expected that some particles have high Stokes numbers, for example due to large diameter and heavy density could directly strike the top of the bottom baffle, and possibly deposit there or slide along the wall of the insert to the bottom of the distributor. That will produce an unpredictable coal slurry distribution. The original idea of having an insert in the distributor might, through this striking process, improve the distribution of the coal slurry, but it doesn't work as well as expected. As an alternative, it might be expected if a spray for the particle phase is set at the top of the distributor instead of the central feed of materials, the particles will have more contact time with the water, the distribution could be improved. In other words, to achieve a better coal slurry distribution, it would be expected that the particle trajectories should be mainly determined by the drag force and be driven around the distributor for a longer particle residence time. Systematically it may be expected that the bottom baffle should be extended into the distributor more, so that all the particles are kept away from the core region and given the maximum mixing velocity, and also the orifice should be removed from the distributor due to a slow motion existing in the lower chamber.

5. CONCLUSIONS

Based on the present computational results, the following conclusions can be drawn:

- the swirling velocity which is proportional to the local radius of the distributor is very important for the coal slurry flow, and a strong circulating motion can result in a longer particle residence time, but in the core region such velocity is very small providing a relatively weak region for the mixing,
- the orifice cannot provide a mixing amplification, instead it reduces the mixing effect in the lower chamber,
- the feeding location of the particulate phase will influence the coal slurry distribution, and off-center feeding can lead better results of distribution.
- the distributor insert has a positive effect on the coal slurry distribution, since it confines all the particles to the relatively high swirling velocity zones,
- the Lagrangian approach can provide a qualitative description for the particulate phase.

It should be noted that the above conclusions are based only on the Lagrangian approach. In the next stage of the work, the Eulerian method will be used to simulate the coal slurry flows. The kinetic theory will be used to model the particulate phase, and the two-equation turbulence model will be modified to account for the particle phase effect on the turbulent structure. Hopefully, detailed computational results will be presented soon.

ACKNOWLEDGMENTS

This project is financially supported by the Australian Postgraduate Award in Industry (APAI) and MIM, which are gratefully acknowledged.

REFERENCES

Chen, C.P. and Wood, P.E., 1985,"A Turbulence Closure Model For Dilute Gas Particle Flows," *Can. J. Chem. Engng.,* 63, pp.349-360.

Ding, J. and Gidaspow, D., 1990, "A Bubbling Fluidization Model Using Kinetic Theory of Granular Flow," *AICh.E.J,* Vol. 36(4), pp.523-538.

Elghobashi, S. E. and Abou-Arab, T. W., 1983,"A Two-Equation Turbulence Model for Two-Phase Flows," *Physics Fluids,* Vol. 26, pp. 931-938.

Fletcher, C. A. J., 1991, "Computational Techniques for Fluid Dynamics I & II," *Springer-Verlag,* Heidelberg.

Fletcher, C. A. J., Jancar, T., Matthews, B. De Guzman, M. M. and Tu, J. Y., 1995, "Computational Simulation for Mineral Processing," *APCOM Conference* XXV, pp. 361-370.

Holtham, P. N., 1995, Personal Communication.

Jenkins, J. T. and Savage, S. B., 1983, "A Theory for the Rapid Flow of Identical, Smooth, Nearly Elastic, Spherical Particles," *J. Fluid Mech.,* vol. 130, pp. 187-202.

Johnson, P. C. and Jackson, R., 1987, "Frictional-Collisional Constitutive Relations for Granular Materials, with Application to Plane Shearing," *J. Fluid Mech.,* vol. 176, pp. 67-93.

Kelly, R. I. and Holtham, P. N. 1996, "An Investigation of Industrial Row Distributor, Part A: Experimental Work," *24th Proceeding Aus. and New Zealand Chem. Eng. Con. and Exhi.,* vol. 3, pp. 23-28.

Morsi, S. A. and Alexander, A. J., 1972, "An Investigation of Particle Trajectories in Two-Phase Flow System," *J. Fluid Mech.*, Vol. 55 (part 2), pp.193-208.

Patankar, S.V., 1980, "Numerical Heat Transfer and Fluid Flow," *Hemisphere Publishing Corporation.*

Partridge, A. C., 1992, "Advanced Coal Preparation Monograph Series," (ed. Swanson, A.R. & Partridge A.C.), *Australian Coal Preparation Society.*

Pitt, G. and Stone, O., 1989, "Raw Coal Distribution Modifications," *Internal Report-German Greek, Ref GRP596/hjm.*

Schwarz, M.P., 1991, "Flow Simulation In Mineral Engineering," *Min. Eng.*, Vol. 4, Nos 7-11, pp.717-732.

Tsuji, Y., Morikawa, Y. and Shiomi, H., 1984,"LDV Measurements of an Air-Solid Two Phase Flow in a Vertical Pipe," *J. Fluid Mech.*, Vol. 139, pp.417-439.

Tu, J. Y. and Fletcher C. A. J., 1995, "Numerical Computation of Turbulent Gas-Solid Particle Flow in a 90^0 Bend," *AICh.Eng. J.*, Vol. 14, No. 10, pp.2187-2197.

Yakhot, V. and Orszag, S. A., 1986, "Renormalization Group Analysis of Turbulence," *I. Basic Theory, J. of Sci. Comput., V.1, No. 1, pp1-51.*

Zhou, Q. and Leschziner, M.A., 1991, *paper presented at the 8th Turbulent Shear Flow Symp., Munich.*

Zisselmar, R. and Molerus, O., 1979, "Investigation of Solid-Liquid Pipe Flow with regard to Turbulence Modification," *Chem. Eng. J.* Vol. 18, pp. 233-239.

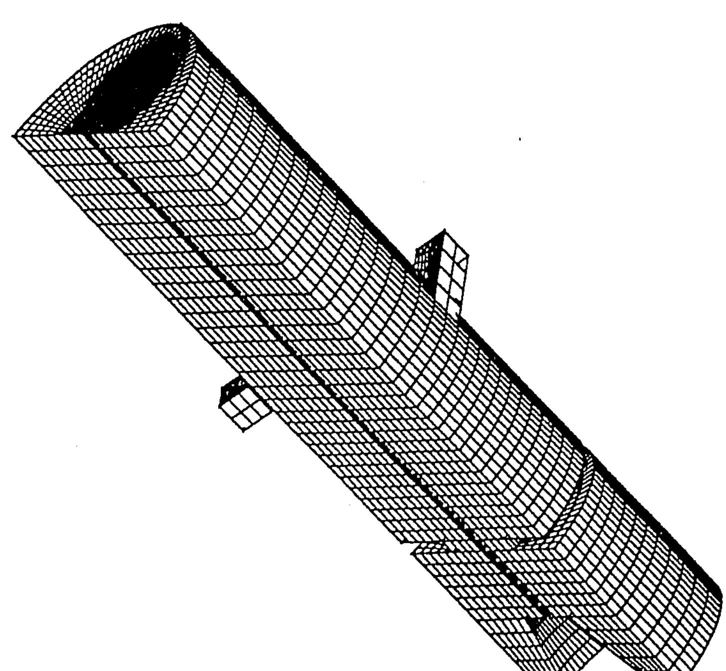

Fig. 1 Computational Mesh

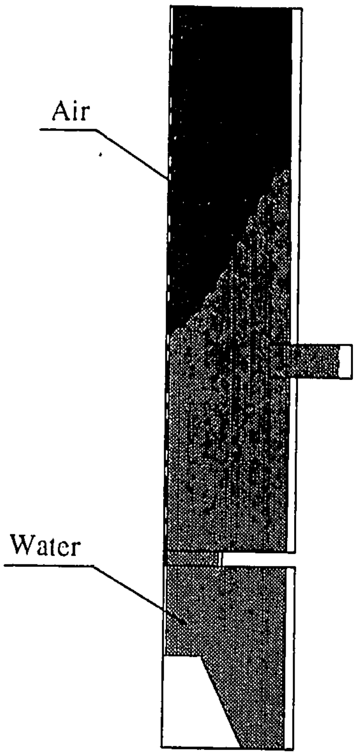

Fig. 2 Volume Fraction

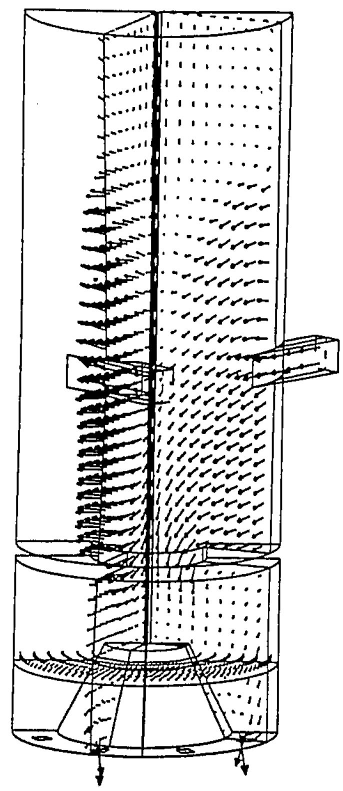

Fig. 3 Velocity Vectors

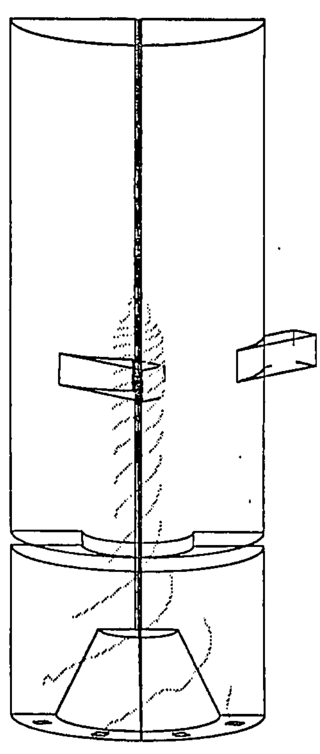

Fig. 4 Centre Feeding Particle Trajectories

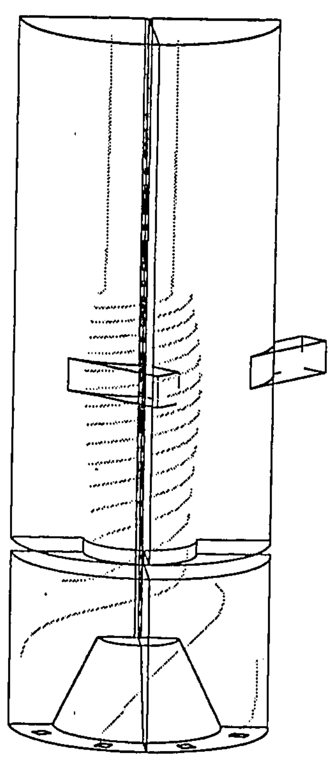

Fig. 5 Off-Centre Feeding Particle Trajectories