PRESSURE DROP PREDICTION IN DENSE PHASE PNEUMATIC CONVEYING USING CFD

Chandana RATNAYAKE[1], Morten C. MELAAEN[1], Biplab K. DATTA[2]

[1] Telemark University College, Kjolnes Ring, 3914 Porsgrunn, NORWAY

[2] Department of POSTEC, Telemark Technological R & D Centre, Kjolnes Ring, 3918 Porsgrunn, NORWAY

ABSTRACT

During the last two decades, computer techniques have become well-accepted and useful tools for studying two-phase gas-solids flow processes. With the development of high speed high-capacity desk top personal computers, scientists and researches have extensively been using several large scale commercial computer codes to predict the flow phenomena in pneumatic conveying systems, which are rather complicated to deal with using only first principle based traditional prediction methods. But, unfortunately, very few research publications could be seen in open literature, which investigated the capabilities of commercially available CFD codes as pressure drop prediction tools in dense phase pneumatic transport systems. In an earlier publication (Ratnayaka et al., 2004), the authors discussed the possibility of prediction of pressure drop across a standard 90º bend using a commercial CFD software package. It was clear that the CFD computer code called Fluent® could be used successfully to calculate the pressure drop across a bend with a considerable degree of accuracy. Only one bulk solid (barite) and one internal pipe diameter (75 mm) was used for the said experimental investigation. This paper presents the results of the experimental and computational study carried out in the second stage of the investigation, which dealt with two more different materials (cement and ilmenite) and different transport line configurations. With this series of tests on different materials and pipe configurations, the earlier findings could be confirmed. With Eulerian granular approach, the predicted pressure drop showed a considerably good agreement with the experimental measurements obtained for a dense phase flow situation with a maximum error margin of ±15%.

INTRODUCTION

The problem of determining the pressure loss produced by a bend has been the subject of research for many years, because of its considerable importance in the design and analysis of pneumatic transport systems. However, no general consensus on how to analyse the two-phase flow across a bend has emerged, since the motion of particles around a bend and exact calculation of the pressure drop caused by a bend are highly complex.

As an alternative approach to the classical methods of analysing gas-solid flows, computational techniques have been a well-accepted tool, throughout the last two decades. Specially, Computional Fluid Dynamics (CFD) technique which means a computational technology that enables one to study the dynamics of flows, has been becoming popular in scientific research field and industry.

CFD involves numerical solution of sets of governing conservation differential equations, and other equations as required. CFD has proven to be extremely useful and accurate for single-phase flow applications. It is now possible to make numerical predictions for many single-phase flows that are more precise than the most accurate experimental local measurements that can be obtained in a physical apparatus of the same geometry. There are two different approaches in numerical models for the gas-solid flow according to the manner in which the particulate phase is treated: Euler-Lagrangian and Euler-Euler granular approaches. Generally, the Lagrangian approach is more suited for dilute flows. It has limited applicability to dense phase flows, because it may be necessary to include an enormous number of particles. On the other hand, the Euler-Euler granular model is suitable for dense gas-solids flows. Some investigations which tried to predict the gas-solid system behaviours using Eulerian approach came with interesting results (Scott, 1978; Manger, 1996; Mathiesen, 1997; Wang, 2001).

In recent years, a break through could be achieved, specially in the CFD applications of dense phase gas-solid flow systems when some research works (Jenkins and Savage, 1983; Lun et al., 1984; Johanson and Jackson, 1987; Sinclair and Jackson, 1989; Ding and Gidaspow, 1990) discussed the dynamics of inter-particle collisions. Based on this concept, Gidaspow (Gidaspow, 1994) explained the kinetic theory approach, which uses one equation model to determine the turbulent kinetic energy of the particles in terms of the granular temperature and assumes either a Maxwellian or non-Maxwellian distribution for the particles applicable to both cases of dilute and dense phases. The kinetic theory approach for granular flow permits the determination of the pressure and viscosity of solids in place of empirical relations for closure of the governing equations of CFD in traditional way of handling. Particularly, when the turbulence is concerned, most of the investigations have focussed on very dilute systems and their applications could most probably be limited to a very dilute flow of particles. In case of dense phase systems, the interaction between gas turbulence and particle fluctuation is significant. Thus, the kinetic theory approach uses an interaction term between gas turbulence and particle fluctuations in the granular temperature equation (Gidaspow, 1994).

Although a large number of research publications of CFD applications on dilute phase pneumatic transport (Yasuna et al., 1995; Mason et al., 1998; Li and Tomita, 2000; Sommerfeld et al., 2001; Srivastava and Sundaresan, 2003) and fluidised bed systems (Mathiesen, 1997; Benyahia et al., 2000; Mathiesen and Solberg, 2004; Ye et al., 2004) can be seen in open literature, the applications

on dense phase conveying is comparatively less. Even if some commercial CFD codes claim better performances in case of dense phase pneumatic conveying applications, their usability in real situations is very limited. On the other hand, only a few cases (Levy et al., 1997; Benyahia et al., 2000; Giddings et al., 2004; Ye et al., 2004) could be found where some attempts were made to use CFD codes to predict the pressure drops across the parts of gas-solid systems. They also concerned either very dilute phase conveying (Levy et al., 1997; Giddings et al., 2004) or fluidised bed applications (Benyahia et al., 2000; Ye et al., 2004). The main objective of this work was to investigate the prediction capability of a commercial CFD code when it is compared with the experimental observations of a dense phase pneumatic conveying system. As an initial step, a standard 90° horizontal to horizontal bend has been studied and the experiments were carried out using three different bulk materials with dense phase flow conditions.

The software program named Fluent® has been chosen to carry out calculations in this particular case. Using the Fluent® version 6 software package, a three-dimensional computational fluid dynamic model has been developed. The simulations have been carried out defining different flow parameters as close as possible to actual experimental conditions. Finally, the comparison and discussion of computational results are presented.

TEST SETUP

A schematic view of the transport line and the considered bend are shown in Figure 1 and 2 respectively. The individual dimensions of bends of different pipe sizes are given in Table 1.
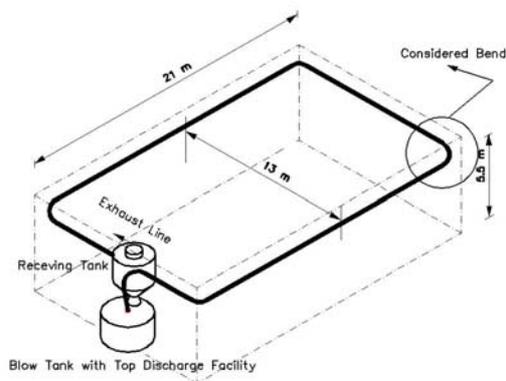
Figure 1: Schematic view of the conveying line
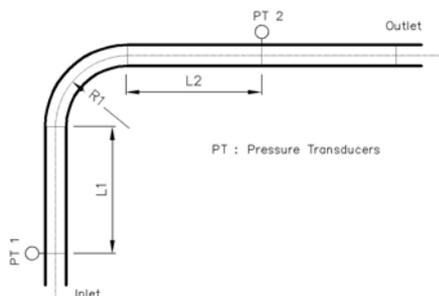
Figure 2: Details of the considered bend

Pipe Diameter (mm)	L1 (mm)	L2 (mm)	R1 (mm)
102	515	600	343
128	515	550	424

Table 1: The dimensions of different bends (nomenclatures in Figure 1)

All signals from the different instruments (pressure transducers, flow transducers etc,) were fed to a desktop computer. The acquisition, sampling and averaging of data are carried out using the LabVIEW® software package. For this investigation, mass flow rate of solids, volume flow rate of air, and pressure readings at inlet and outlet of considered bend have been recorded. The details of experimental setup have been published elsewhere (Ratnayaka et al., 2004).

MATERIAL PROPERTIES

Two different materials were used for the investigation and the properties of those are shown in Table 2.

Test Material	Mean Particle Size (µm)	Particle Density (kg/m^3)
Ilmenite	9.5	4600
Cement	15.5	3100

Table 2: Data for test materials

THEORETICAL CONSIDERATION

In CFD analysis technique, a single set of conservation equations for momentum and continuity is usually solved for a single-phase system. But, for multi-phase flow applications, obtaining accurate solutions is much more challenging, not just because each of the phases must be treated separately, but, in addition, a number of new and difficult factors come into play. Some of these additional factors, which are intended to accommodate the inter-phase interactions in a multiphase system, can be listed as follows;

- Drag and lift forces
- ''Slip'', i.e. relative motion, between the phases
- Electrostatic and/or electrophoretic forces
- Particles sizes, size distribution and shapes
- Inter-particle forces
- Inter-particle collisions
- Collisions/interactions of particles with the wall of the containing vessel.

As a result of such factors, the CFD models for dense multiphase systems have adopted a wide range of different approaches: Eulerian, Lagrangian or combinations of the two, interpenetrating two-fluid approach or discrete particles, two-dimensional or three dimensional codes, solution via finite difference, finite element or finite volume approaches, etc.

In the process of introducing additional sets of conservation equations, the original set must also be modified. The introduction of the volume fractions of gas and solid phases and the exchange mechanism for the momentum between the phases are the main modifications for a multi-phase CFD model.

As mentioned earlier, Fluent® 6.1 was used to simulate the flow across the considered bend. The Euler-Euler granular multiphase model, which considers the solid and gas

phases as interpenetrating continuum that share the space, has been used for this simulation. Generally, Fluent® uses a control-volume based technique to convert the governing differential equations of the flow system to algebraic equations which can be solved numerically. The volume averaged dicretization approach consists of integrating the governing equations over each and every control volume, generating separate equations that conserve each quality on a control-volume basis. The discretized equations, along with the initial and boundary conditions, were solved to obtain a numerical solution. Conservation equations of mass and momentum were developed using the Eulerian approach, and solved simultaneously by considering the phases separately but linking them through the drag forces in the momentum equation. In the Fluent® software code, the Phase Coupled SIMPLE (PC-SIMPLE) algorithm which is an extension of the SIMPLE algorithm (Patankar, 1980) to multiphase flows is used for the pressure-velocity coupling. The velocities are solved coupled by phases, but in a segregated fashion. A pressure correction equation is built based on total volume continuity rather than mass continuity. Pressure and velocities are then corrected so as to satisfy the continuity constraint. To model the turbulence, the per phase k-ε (k: turbulent kinetic energy, ε: turbulent kinetic energy dissipation) model has been used. The full details of the model and the solution procedure are given in detail in the Fluent® user's guide (2003).

EXPERIMENTAL PROCEDURE

The experiments were carried out to obtain the pneumatic conveying characteristics for barite, ilmenite and cement using three different pipeline configurations of different bore sizes, in terms of air mass flow rate, mass flow rate of material and total pressure drop in the conveying line. Initially, the desired value of blow tank pressure was set using the pressure control switch on the control panel. After that, the blow tank was filled with the test material. Then, the blow tank was pressurised by opening the supply air valve. When the blow tank pressure reached the set value of transport pressure, the material was introduced to the conveying line, by opening the discharge valve. At the same time, the data logging system of the computer was started. The variation of pressure signals at desired points on the conveying line and weight accumulation of the receiving tank were graphically indicated on the computer. After each test run, the relevant data were obtained by averaging during the specified time interval during which the stable conveying conditions were detected. Further details of the experimental procedure is published in early publications by the authors (Datta and Ratnayaka, 2003; Datta et al., 2003; Ratnayaka et al., 2004).

NUMERICAL SIMULATION

A three-dimensional grid has been prepared using body-fitted coordinate technique, with the Gambit® software package according to the actual dimensions of the considered bend. At the inlet, the calculation domain starts exactly at the position of the inlet pressure transducer. To ensure the steady flow conditions at the outlet of calculation domain, the grid was extended by 0.5 m after the outlet pressure transducer (refer Figure 2). A typical

grid of the calculation domain relevant to 102 mm diameter pipe is shown in Figure 3.
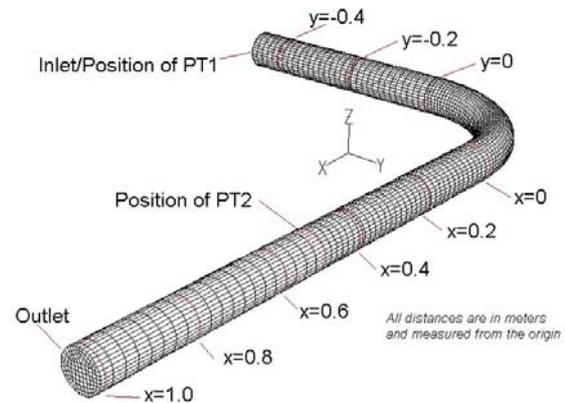
Figure 3: A typical grid of the calculation domain of 102mm diameter bend.

As shown in Figure 3, few surfaces were generated within the pre-bend and the post-bend pipe sections for the purpose of using them in post processing as reference planes, to check the variation of solid concentration.

The grid files generated by Gambit® were used for the simulation in Fluent®. The different simulation parameters have been defined as close as possible to the actual experimental conditions.

In the problem specification, the discretization method called 'Phase Coupled SIMPLE' was selected for the pressure-velocity coupling while 'First Order Upwind' discretization method was used for other scalar parameters like momentum, volume fraction, turbulent kinetic energy, etc. The simulations have been carried out for different test conditions in terms of air flow rate, material mass flow rate and total pressure drop value. Inlet boundary conditions such as inlet pressure, volume fraction of solid, etc, have been defined according to the experimental inlet condition of the bend. At the wall, a no slip condition is assumed and the wall roughness constant was taken as 0.5. For the convenience of the simulations, spherical mono sized particles were assumed for all bulk materials and particle mean diameters (d_{50}) were used for the simulations. To define the boundary conditions at the bend inlet, the velocities of both phases have to be given. The air velocity was determined according to the experimental measurements, while the concept of slip velocity had to be considered to calculate the solid phase velocity. Although a considerable number of models to calculate the slip velocity in dilute phase could be seen in open literature (Sankar and Smith, 1986; Sankar and Smith, 1986; Lodes and Mierka, 1989; Raheman and Jindal, 1993; Li and Tomita, 2002), a very few models (Midttveit and de Silva, 1990; Hong and Shen, 1993) is available for the same in dense phase. To estimate the solid velocity, a graphical presentation of slip velocity curves (Woodcock, 1987) which is based on the model proposed by Arastoopour et al. (Arastoopour et al., 1979) was used. The volume fraction of solids at the bend inlet was calculated according to the experimental condition.

RESULTS AND DISCUSSION

During the experimentations, special attempts were made to cover low air velocity regions of conveying.

Consequently, the conveying data were of high solid loading ratios (the ratio between mass flow rate of solids and mass flow rate of air), ranging from 29 to 257. The conveying air velocities and the calculated volume fraction of solids at the bend inlet were approximately in range of 4 - 17 m/s and 9%-41% respectively.

The Fluent® provides with the facilities to post-process the data and display the necessary contour, vector and profile plots of pressures, velocities, volume fractions, etc, of individual phases. After achieving the convergence condition, contour plots of solid phase were examined for each simulation. Using these plots, the characteristics of the flow across the pipe bend could be understood. In particular, the plots of contours of volume fraction are showing the flow behaviour of each phase which is very difficult to observe during the experimentations. In order to conserve the length of this paper, only representative results of the numerical simulations are given here.

The results are categorised into two groups and presented here. Firstly, the variation of solid volume fraction is observed within the calculation domain while the pressure drop determination and comparison with experimental data are given at the end.

Variation of volume fraction of solid phase

After reviewing the contour plots of volume fractions of gas and solid phases, it could be noticed that all the plots showed a quite similar behaviour, especially in case of their propagation across the bend, independent from conveying bulk materials and pipe diameters. One such contour plot of variation of cement volume fraction across the 102 mm bend is shown in Figure 4.
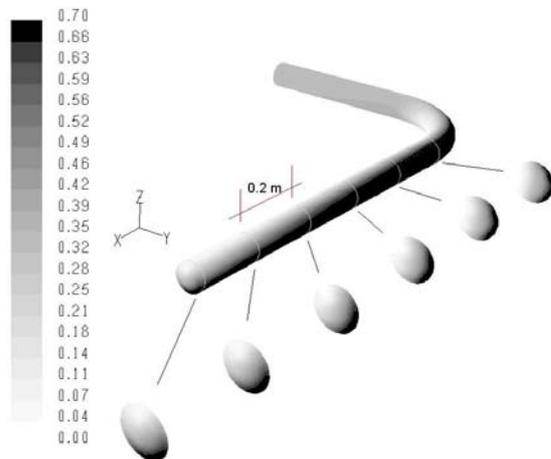
Figure 4: Contours of volume fraction (Material: cement, Diameter: 102 mm)

Figure 4 depicts the contour plot of solid phase volume fraction of bend and few cross sections along the axis of the pipe at 200 mm intervals in down stream section of the bend. Figure 3 shows the exact positions of the surfaces. The contour plots clearly show a high particle concentration at the long radius wall of the bend. In the vicinity of outlet, similar high particle concentration could be seen at the bottom area of the conveying pipe. With the help of cross sectional contour plots, one can obviously see that the area of high particle concentration slides gradually from the long radius side wall to the lower half of the pipe along the pipeline as the gas-solid mixture is heading out of the bend. This explains the behaviour of the solid particle inside the bend, specially, at the middle

section of the bend where the solid particles are subjected to the centrifugal forces and thrown towards the pipe wall. Once the centrifugal action starts diminishing as the flow moves towards downstream side of the bend, the effect of gravitational force becomes more dominant and the particles tend to concentrate at the pipe bottom giving high solid volume fraction in the lower half of the pipe cross-section.

Similar phenomenon was reported by many other investigators (Huber and Sommerfeld, 1994; Huber and Sommerfeld, 1998; Levy and Mason, 1998; Venkatasubramanian et al., 2000; Bilirgen and Levy, 2001; Zhu et al., 2003) as well. Most of them explained the presence of high solid fraction at the bottom of the horizontal pipe wall as a result of gravitational settling of solids. Huber and Sommerfeld (Huber and Sommerfeld, 1994) described it as a result of high loss of solids momentum at the bottom of the pipe due to more intense particle-particle and particle-wall collisions as compared to the upper portion of the pipe. They also found inelastic particle collisions and viscous dissipation of the gas phase as the reasons for high momentum loss. Some research workers (Levy and Mason, 1998; Yilmaz and Levy, 1998; Bilirgen and Levy, 2001; Giddings et al., 2004) have observed a characteristics of a rope of solid particles inside and/or just downstream of a bend. Some of them also reported a secondary flow pattern of a double vortex at the downstream of the bend. However, these phenomena are quite common in dilute phase conveying where the solid particles have more freedom to move. But, the flow modes under the consideration of this investigation were completely in dense phase (the solid loading ratio ranging from 29 to 257). Consequently, the solid particles have less freedom to move around the conveying domain, thus the roping phenomenon may not be applicable here.

Pressure drop determination

During the post processing of simulation data, it was possible to determine the pressure drop across the considered bend. The pressure drops across the bend, mainly in between the locations of two pressure transducers, were calculated with the help of the simulation results and compared with the experimental observations. Figure 5 shows the graph of experimental pressure drop values vs. calculated pressure drop results for two conveying materials for three different pipe sizes.
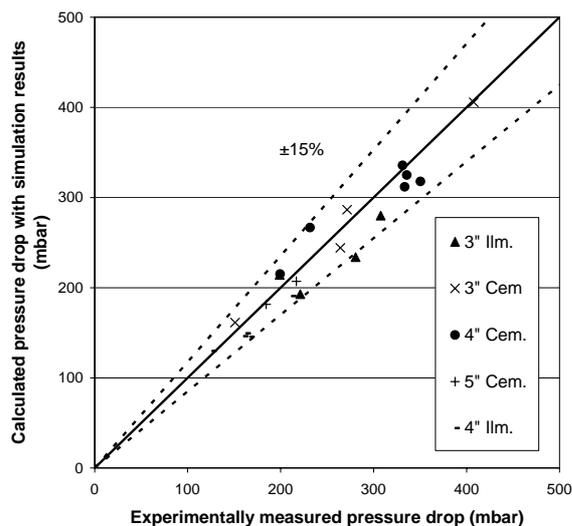
Figure 5: Simulation vs. experimental results of pressure drop across the bend

According to Figure 5, it is clear that the simulation results are in considerably good agreement with the experimental observation, with ±15% deviation in general. This is a quite interesting result, particularly without using any user defined subroutines or any such additional tuning of the basic model available in Fluent® 6, except for careful selection of the boundary conditions.

As mentioned earlier, it was assumed that the solid particles are of mono-size and spherical in shape. This assumption deviates from the real situation where cement and ilmenite are roughly of cubical and conchoidal shapes respectively. Similarly, the assumption of mono-sized particles is also not compatible with the real situation. Even with these assumptions, the pressure drop calculations were within acceptable range with basic model formulation available in the Fluent® software package. With this observation, it is clear that the Fluent® software code can reliably be used to calculate and predict the flow phenomena in dense phase pneumatic transport.

CONCLUSION

The Euler-Euler granular approach with the kinetic theory for the multiphase flow has been used to simulate the gas-solid particle flow in three-dimensional standard 90° horizontal to horizontal bend in a pneumatic transport system with dense phase flow condition, using a commercial CFD software package; Fluent®. To simulate the turbulence for the gas and solid phases, the per phase k-ε model has been used. Pneumatic conveying tests were carried out using two different materials and three different pipeline configurations with respect to the pipe diameter.

The results on variations of volume fraction of solid particles, gas and solid phase velocity profiles and pressure drop determination across the bend were analysed using the post-processing facilities available with the software. Although no experimental measurement was made on solid particle concentrations, a good quantitative agreement of the simulation results could be detected with the published results by other research workers. In case of pressure drop determination, considerably good agreement could be found between the experimental measurements and simulation results with ± 15% error margin for all

conveying conditions and combinations under the consideration of this investigation.

From the simulation results, it is clear that Fluent® CFD software has a good potential to model the gas-sold flow across a bend in dense phase pneumatic conveying satisfactorily. In fact, this investigation and the simulation work have been carried out as an initial step towards a reliable modelling technique of dense phase pneumatic conveying.

REFERENCES

Fluent 6.1 User's Guide (2003), Fluent Inc., Lebanon.

ARASTOOPOUR, H., MODI, M.V., PUNWANI, D.V. and TALWALKAR, A.T., (1979). "A Review of Design Equations for the Dilute-Phase Gas-Solids Horizonntal Conveying Systems for Coal and Related Materials". *Proc. of International Powder and Bulk Solids Handling and Processing Conference*, Philadelphia.

BENYAHIA, S., ARASTOOPOUR, H., KNOWLTON, T.M. and MASSAH, H., (2000), "Simulation of particles and gas flow behavior in the riser section of a circulating fluidized bed using the kinetic theory approach for the particulate phase", *Powder Technology*, **112**.(1-2), 24-33.

BILIRGEN, H. and LEVY, E.K., (2001), "Mixing and dispersion of particle ropes in lean phase pneumatic conveying", *Powder Technology*, **119**.(2-3), 134-152.

DATTA, B.K. and RATNAYAKA, C., (2003), "A simple technique for scaling up pneumatic conveying systems", *Particulate Science and Technology*, **21**.(3), 227-236.

DATTA, B.K., RATNAYAKA, C., SAASEN, A. and BASTESEN, Y., (2003), "A New Design Approach for of Pneumatic Conveying", *Annual Transactions of the Nordic Rheology Society*, **11**, 57-62.

DING, J. and GIDASPOW, D., (1990), "A Bubbling Fluidization model using kinetic theory of granular flow", *Aiche Journal*, **36**, 523-538.

GIDASPOW, D., (1994), "Multiphase Flow and Fluidization: Cotinuum and Kinetic Theory Description", Academic Press.

GIDDINGS, D., AROUSSI, A., PICKERING, S.J. and MOZAFFARI, E., (2004), "A 1/4 scale test facility for PF transport in power station pipelines", *Fuel*, **83**.(16), 2195-2204.

HONG, J. and SHEN, Y., (1993). "Characteristics in Horizontal Pneumatic Conveying at High Solids Loading and Low Gas Velocity". *International Symposium on reliable Flow of Particulate Solids II*, Oslo, Norway, POSTEC A/S.

HUBER, N. and SOMMERFELD, M., (1994), "Characterization of the Cross-Sectional Particle Concentration Distribution in Pneumatic Conveying Systems", *Powder Technology*, **79**.(3), 191-210.

HUBER, N. and SOMMERFELD, M., (1998), "Modelling and numerical calculation of dilute-phase pneumatic conveying in pipe systems", *Powder Technology*, **99**.(1), 90-101.

JENKINS, J.T. and SAVAGE, S.B., (1983), "A Theory for the rapid flow of identical, smooth, nearly elastic spherical particles", *Journal of Fluid Mechanics*, **176**, 67-93.

JOHANSON, P.C. and JACKSON, R., (1987), "Frictional-collision constitutive relation for granular materials, with application to plane shearing", *Journal of Fluid Mechanics*, **176**, 67-93.

LEVY, A. and MASON, D.J., (1998), "The effect of a bend on the particle cross-section concentration and segregation in pneumatic conveying systems", *Powder Technology*, **98**.(2), 95-103.

LEVY, A., MOONEY, T., MARJANOVIC, P. and MASON, D.J., (1997), "A Comparison of Analytical and Numerical Models with Experimental Data for Gas-Solid Flow through a Straight Pipe at Different Inclinations", *Powder Technology*, **93**.(3), 253-260.

LI, H. and TOMITA, Y., (2000), "A numerical simulation of swirling flow pneumatic conveying in a horizontal pipeline", *Particulate Science and Technology*, **18**.(4), 275-291.

LI, H. and TOMITA, Y., (2002), "Measurements of particle velocity and concentration for dilute swirling gas-solid flow in a vertical pipe", *Particulate Science and Technology*, **20**.(1), 1-13.

LODES, A. and MIERKA, O., (1989), "particle velocities in two-phase solid-gas flow", *Powder Technology*, **58**, 163-168.

LUN, C.K.K., SAVAGE, S.B., JEFFREY, D.J. and CHEPURNIY, N., (1984), "Kinetic Theories for Granular Flow: Inelastic Particles in Couette Flow and Slightly Inelastic Particles in a General Flow Field", *Journal of fluid mechanics*, **140**, 223-256.

MANGER, E., (1996), "Modelling and Simulation of Gas/Solids Flow in Curvilinear Coordinates", Ph.D. Thesis, Process Technology, Telemark University College, Norway.

MASON, D.J., MARJANOVIC, P. and LEVY, A., (1998), "A simulation system for pneumatic conveying systems", *Powder Technology*, **95**.(1), 7-14.

MATHIESEN, V., (1997), "An Experimental and Computational Study of Multiphase Flow Behaviour in Circulating Fluidizing Beds", Ph.D. Thesis, Process Technology, Telemark University College, Norway.

MATHIESEN, V. and SOLBERG, T., (2004), "Laser-Based Flow Measurements of Dilute Vertical Pneumatic Transport", *Chemical Engineering Communications*, **191**.(3), 414-433.

MIDTTVEIT, Ø. and DE SILVA, S.R., (1990). "Slip Velocity in Pneumatic Conveying". *4th International Conference on Pneumatic Conveying Technology*, Glasgow, Scotland, powder Advisory Centre, UK.

PATANKAR, S.V., (1980), "Numerical Heat Transfer and Fluid Flow", Hemisphere Publication Corp., Washington.

RAHEMAN, H. and JINDAL, V.K., (1993), "Slip velocity in pneumatic conveying of agricultural grains", *Powder handling & processing*, **5**.(1), 15-20.

RATNAYAKA, C., MELAAEN, M.C. and DATTA, B.K., (2004), "An Experimental Investigation and a CFD Modelling of Gas-Solid Flow across a Bend in a Dense Phase Pneumatic Conveying System", *Proc. of 8th International Conference on Bulk Materials Storage, Handling and Transportation*, 388-393.

SANKAR, S.R. and SMITH, T.N., (1986), "Slip Velocities in Pneumatic Transport.1", *Powder Technology*, **47**.(2), 167-177.

SANKAR, S.R. and SMITH, T.N., (1986), "Slip Velocities in Pneumatic Transport.2", *Powder Technology*, **47**.(2), 179-194.

SCOTT, A.M., (1978), "Pneumatic Transport of Granules", *Chemical Engineer-London*.(330), 189-190.

SINCLAIR, J.L. and JACKSON, R., (1989), "Gas-particle flow in a verticle pipe with particle-particle interactions", *AIChE Journal*, **36**.(4), 523-538.

SOMMERFELD, M., LAIN, S. and KUSSIN, J., (2001). "Analysis of Transport Effects of Turbulent Gas-Particle Flow in a Horizontal Channel". *Proc. of 4th International Conference on Multiphase Flow*, New Orleans, ICMF.

SRIVASTAVA, A. and SUNDARESAN, S., (2003), "Analysis of a frictional-kinetic model for gas-particle flow", *Powder technology*, **129**.(1), 72-85.

VENKATASUBRAMANIAN, S., TASHIRO, H., KLINZING, G.E. and MYKELBUST, K., (2000), "Solids Flow Behaviour in Bends: Assessing Fine Solids Buildup", *Powder Technology*, **113**, 124-131.

WANG, Q., (2001), "An Experimental and Computational Study of Gas/Particle Multiphase Flow in Process Equipment", Telemark University College. Norway.

WOODCOCK, C.R., MASON, J.S., (1987), "Bulk Solids Handling - An Introduction to the Practice and Technology", Leonard Hill.

YASUNA, J.A., MOYER, H.R., ELLIOTT, S. and SINCLAIR, J.L., (1995), "Quantitative Predictions of Gas-Particle Flow in a Verticle Pipe with Particle-Particle Interactions", *Powder Technology*, **84**, 23-34.

YE, M., VAN DER HOEF, M.A. and KUIPERS, J.A.M., (2004), "A Numerical Study of Fluidization Behaviour of Geldart A Particle using a Discrete Particle Model", *Powder Technology*, **139**, 129-139.

YILMAZ, A. and LEVY, E.K., (1998), "Roping phenomena in pulverized coal conveying lines", *Powder Technology*, **95**.(1), 43-48.

ZHU, K., MADHUSUDANA RAO, S., WANG, C.-H. and SUNDARESAN, S., (2003), "Electrical capacitance tomography measurements on vertical and inclined pneumatic conveying of granular solids", *Chemical Engineering Science*, **58**.(18), 4225-4245.