The Canadian Society for Bioengineering
The Canadian society for engineering in agricultural, food, environmental, and biological systems.

La Société Canadienne de Génie Agroalimentaire et de Bioingénierie
La société canadienne de génie agroalimentaire, de la bioingénierie et de l'environnement

CSBE | SCGAB

Paper No. CSBE17-009

Simulation of Multiphase Flow Conditions in Air Seeders for Control Applications

Jason D.M. Cousins

Department of Mechanical Engineering, University of Saskatchewan, Saskatoon, SK, CANADA.

Scott D. Noble

Department of Mechanical Engineering, University of Saskatchewan, Saskatoon, SK, CANADA.

Written for presentation at the
CSBE/SCGAB 2017 Annual Conference
Canad Inns Polo Park, Winnipeg, MB
6-10 August 2017

ABSTRACT Air seeders distribute seed and granular fertilizers to numerous tillage implements through a distributed pneumatic conveying system. Real-time prediction of flow within the conveying lines could be used to improve the controllability and performance characteristics of existing air seeder systems. The purpose of this study was to develop a model and simulation for the flow conditions found on air seeders, and to evaluate its accuracy and applicability to control application. Computer simulation has been applied to prediction of the gas-particle flow in a straight section of an air seeder primary conveying line. A low-computational cost, one-dimensional computational fluid dynamics approach is used. The model is constructed using an Eulerian-Eulerian (two-fluid) modeling framework. Discretization of the equations is conducted using the finite volume method and solution of the discrete equations follows the SIMPLER technique. Simulated data for particle velocity and fluid pressure were compared to experimental steady-state measurements. Simulation results for fluid pressure were found to have good agreement with experimental data, with an average percent error of 3.6% and a maximum value of 14.1%. Predicted solids velocities were found to have poor agreement, with an average percent error of 18.7%, and a maximum value of 39.3%. However, further investigation into elements of the model (e.g. drag force estimation) could potentially improve accuracy of the simulation. Furthermore, computing time was found to scale favorably vs. accuracy as grid resolution was varied. Overall computing times observed for the simulation indicate strong potential for real-time control application.

Keywords: simulation, control, pneumatic conveying, multiphase flow, CFD, air seeder

---

INTRODUCTION Air seeders are a common and important agricultural implement used in seed planting and tillage processes. They allow for seed placement, fertilization and soil tillage to be completed in a single operation, using a single piece of equipment. Air seeders store seed and granular fertilizers in a central tank located on the air cart. Products are transported from the air cart through a pneumatic conveying network to a large number of tillage implements located on the air hoe drill. Since its development in the 1950's, air seeding has become a significant part of many conservation tillage operations. The technology has helped to limit soil degradation caused by excess tillage, improving sustainability of the global agricultural food supply.

Despite continual development, there is still a demand for improved performance of modern air seeding equipment. Performance limitations of the pneumatic conveying system represent a significant obstacle to further improvement and increased precision of air seeding technology. For this reason, improving performance of the conveying system is an active area of research for air seeder manufacturers. Typical air seeder conveying systems make use of a single hydraulically powered fan to generate the air flow required to convey product. Reliance on a single air supply introduces coupling between the multiple conveying lines used to distribute product, thus their operation is not independent of one another. To avoid blockages created by inadequate air flow, current practice is to run air speeds far in excess of what is required to convey in most lines. High conveying speed uses more of the tractor's hydraulic capacity, increases energy consumption, increases product damage and reduces germination rates. Variable rate and sectional control features often amplify these deficiencies due to the tight coupling between conveying lines.

Addition of further control mechanisms to the conveying system represents a potential means of improving the performance of air seeding technology. Improved control features would allow for separate conveying lines to be regulated individually, something not possible through control of the shared air supply. Independent control of the conveying lines could allow for the conveying system to be actively balanced, optimizing conveying performance. However, the complexity of the conveying system makes implementation of traditional feedback control difficult. Feedforward control may be a more practical approach, but requires knowledge of the system's operation to be realized. Modeling and simulation is a prospective method for providing the information needed for more advanced control strategies.

There is a long history of modeling related to pneumatic conveying and more general fluid-particle flows. The subject has accumulated a vast body of literature, focused on a wide variety of applications and methods. Many of the available methods involve numerical computation of complex differential equation models; relevant examples of such include the work of Sarrami Foroushani & Nasr Esfahany (2015), Eskin, Leonenko, & Vinogradov (2007); Laín & Sommerfeld (2012) and Andrews & O'Rourke (1996). However, there is a noticeable gap in the literature regarding studies with specific relevance for the specific application of interest herein. Air seeder pneumatic conveying involves very dilute fluid particle flow, with relatively large, high inertia particles, spread throughout a distributed conveying network. Research examining cases with all three of these traits is quite rare. Furthermore, much of the available research focuses on large, complex 3D simulations. While often capable of high accuracy, these approaches are far too computationally intensive to provide information at realistic speeds for control application. This research seeks to fill a niche void in the existing body of literature where little directly applicable work has been done previously.

To provide information for control via modeling and simulation, a suitable model and numerical strategy must be selected. A large number of different strategies for multiphase flow modeling have been successfully employed in the literature. Selection of the most appropriate method will depend on the specifics of the problem being considered. In the present case – the intended application being real-time control – the speed that information is delivered becomes of significant importance. An appropriate method therefore should be as computationally inexpensive as possible, while still providing a moderate degree of accuracy for a limited number of bulk flow quantities. The exact microscopic details of the flow are unlikely to be relevant from a control perspective and so bulk description of the flow is suitable. Furthermore, the dynamic characteristics of the conveying system may be significant for the intended application. This excludes steady-state models, correlations and empirical models and limits selection to those methods based on the numerical solution of differential equations.

A computational fluid dynamics approach was taken in the present research, using numerical solution of partial differential governing equations to predict flow conditions within the conveying lines. A one-dimensional representation of the bulk multiphase pipe flow was used in order to improve computational efficiency at the expense of detail. An Eulerian-Eulerian or “two-fluid” framework was used to represent the fluid and particles (solids phase). Simplifications to the governing equations were made where possible to further improve computational efficiency and simplicity. Discretization of the governing equations for numerical solution was conducted using the finite volume approach. Solution of the equations was based upon the SIMPLER algorithm (Patankar 1980), with some modification for inclusion of the solids phase. Implementation of the above model and numerical strategy was conducted using MatLab.

The objective of the present study was to develop a model and simulation for prediction of multiphase flow conditions found on air seeders, which is suitable for control application. There are several goals for completion of the stated objective. First, selection of a suitable model for representing the multiphase pneumatic conveying flow. Second, to develop a computer simulation program for solving the relevant equations numerically. Third, to evaluate the accuracy of the simulated results using experimental data.

THEORETICAL SECTION

Mathematical Model For reasons described previously, the Eulerian-Eulerian framework was used herein to assemble the multiphase flow model. Commonly referred to as the “continuum” or “two-fluid” approach, the Eulerian-Eulerian framework is one of several different approaches for applying physical laws to develop a mathematical description of the flow. In this method both phases are modeled as continuous mediums, and thus a similar analysis is made of both the fluid and particulate phase. To derive the modeling equations, conservation laws for mass, momentum and energy are applied over a fixed control volume separately to each phase to derive two sets of partial differential equations. Collective treatment of the particles yields fluid-like equations describing a “solids” phase. The fluid and solids phases, each with their own set of governing equations, are coupled through interface equations representing any transfer of mass, momentum or energy between the two phases.

Application of the conservation laws produces a large set of governing equations which describe the behavior of each phase. Introducing two important assumptions, the equations set can be reduced substantially. First, it was assumed that a simple, one-dimensional representation of the flow is suitable to capture the properties and behavior of interest for the given application. Second, temperature variation within the system being analyzed was assumed to be insignificant, allowing for the equation for energy conservation to be discarded. Remaining for each phase were a continuity equation – governing conservation of mass – and a momentum balance equation for the axial (x) or streamwise direction.

Additional assumptions were made to simplify the fluid phase governing equations. The fluid phase, in this case air, was assumed to be incompressible and therefore of constant density. This simplification removed the connection between pressure and density, and simplified the solution of the equations. Justification for incompressible flow includes relatively low pressure and temperature variation under normal operation. Furthermore, the air has far faster dynamics than the solids phase and was therefore assumed to have little influence on overall system dynamics. Fluid shear stresses were approximated as pipe wall friction and were incorporated in the source terms of the momentum equation. With the given assumptions, the final simplified form of the continuity and momentum equations which was solved for the fluid phase are, respectively:

$$\rho_g \frac{\partial}{\partial t}\left(\alpha_g\right) + \rho_g \frac{\partial}{\partial x}\left(\alpha_g u_g\right) = 0 \tag{1}$$

$$\rho_g \frac{\partial}{\partial t}\left(\alpha_g u_g\right) = \ -\rho_g \frac{\partial}{\partial x}\left(\alpha_g u_g^2\right) - \alpha_g \frac{\partial P}{\partial x} + \frac{\partial}{\partial x}\left(\mu \frac{\partial u_g}{\partial x}\right) + S_g \tag{2}$$

The two equations for the fluid phase, continuity (1) and momentum (2) were solved for the two dependent variables: mean fluid velocity (u_g) and fluid pressure (P).

Similarly, further assumptions were made to simplify the equations for the solids phase. Density was taken to be constant, which for the solids is more fact than assumption. Shear stresses were once again moved into the source term and approximated as pipe wall friction. The final assumption was that particle-particle interactions within the solids phase were negligible and could be ignored. While not strictly true, the very dilute nature of the flow implies that particle-particle collision frequency is relatively low. Thus the significance of inter-particle forces is lower than in higher particle concentration flows, where such forces can be relevant (Laín and Sommerfeld 2012). With the given assumptions, the final simplified form of the continuity and momentum equations which were solved for the solids phase are, respectively:

$$\rho_s \frac{\partial}{\partial t}\left(\alpha_s\right) + \rho_s \frac{\partial}{\partial x}\left(\alpha_s u_s\right) = 0 \tag{3}$$

$$\rho_s \frac{\partial}{\partial t}\left(\alpha_s u_s\right) = \ -\rho_s \frac{\partial}{\partial x}\left(\alpha_s u_s^2\right) + \frac{\partial}{\partial x}\left(\mu_s \frac{\partial u_s}{\partial x}\right) + S_s \tag{4}$$

For the solids phase, dependent variables to be solved for include the void ratio (i.e. particle concentration) (α_s) and mean solids velocity (u_s). The final term in the momentum equations for both phases, S, represents the grouped source terms which account for generation and destruction effects. Momentum transfer between the two phases, e.g. drag force, was also accounted for in the source terms.

In addition to the four governing equations (1-4) presented above, there were several auxiliary equations required to complete the mathematical model of the flow. The auxiliary equations were used to calculate required unknown quantities in the governing equations, particularly the momentum source terms (S). For example, friction caused by the pipe wall for each phase must be approximated and introduced in the momentum source terms. For simplicity and efficiency, fluid wall friction was approximated by means of the commonly used Colebrook equation, given as:

$$\frac{\Delta P}{\Delta x} = f \frac{\rho {u_g}^2}{2d}\ , \qquad \frac{1}{\sqrt{f}} = -2\, log_{10}\left(\frac{\varepsilon}{3.7 d_h} + \frac{2.51}{Re\sqrt{f}}\right) \tag{5}$$

The approach of Eskin, Leonenko, & Vinogradov (2007) was used for approximating friction losses in the solids phase. The method assumes that axial momentum loss is primarily due to particle collisions with the wall. Particle collisional frequency and momentum loss per collision were used in conjunction to estimate momentum loss in the axial direction for the bulk flow as:

$$F = (e-1)\frac{4}{3}\frac{\varepsilon \rho_s u_s}{d}\sqrt{{u_s}^2/_{2\pi}} \qquad (6)$$

Momentum exchange between the fluid and solids phase occurs via drag force imparted by the fluid on the suspended particles. Drag force is the primary source of coupling between the phases, with voidage ratio, ($\alpha_g = 1 - \alpha_s$), being the other less significant factor. Bulk drag force between the fluid and solids phases is calculated using the model of Gidaspow (1994), as:

$$F = \beta A\left(u_g - u_s\right), \quad \beta = \frac{3}{4} C_D \frac{\alpha|u_g - u_s|\rho_g(1-\alpha)}{d_p}\alpha^{-2.65} \qquad (7)$$

It should be noted that equation (7) is for fully developed flow, and may provide poor accuracy for the regions of the flow where significant solids acceleration occurs. Other potential substitutes do exist and may need to be investigated in the future. The final quantity needing to be estimated was the solids shear viscosity, μ_s, which appears in the solids momentum transport equations. Solids bulk viscosity was given as (Sarrami Foroushani & Nasr Esfahany 2015):

$$\mu_s = \frac{4}{5}\alpha_s^2 \rho_s d_s (1-e)\sqrt{{u_s}^2/_{3\pi}} \qquad (8)$$

Equations (1) – (8) represent the complete mathematical description of the flow. Numerical solution of the relevant governing equations was then conducted.

Numerical Methodology With the mathematical model defined, a suitable numerical methodology for solution of the equations was selected. The physical system being modeled – multiphase pipe flow – was discretized using a uniform, one-dimensional grid aligned in the axial direction. Discretization of the partial differential equations was conducted using the finite volume method and a staggered grid for separation of the scalar and vector flow variables. An upwind scheme – variables at control volume boundaries are inherited from the upwind control volume – was used for the convective terms. The result was a set of discrete algebraic equations which replaced the governing equations previously presented. Application of the discrete equations to each control volume within the computational domain produced several sets of discrete equations. After subsequent linearization of the source terms, these systems of discrete equations were solved using linear algebra to determine an approximate solution of the flow field.

In the solution of fluid flow problems, tight coupling between pressure and velocity can lead to difficulty in obtaining a solution to the equations. Special techniques are often required to ensure that the numerical solution converges. The solution procedure employed herein was based on the SIMPLER technique developed by Patankar (1980) and modified for inclusion of the solids phase. SIMPLER, an acronym for Semi-Implicit Method for Pressure Linked Equations Revised, is an algorithm designed for solving the tightly coupled governing equations found in fluid flow problems. The SIMPLER algorithm makes use of several intermediate equations to assist in convergence of the fluid velocity and pressure solutions. Successive iteration improves the approximate solutions until a suitable level of convergence has been obtained. The same calculation procedure can be used without modification for both steady-state and time dependent problems.

RESULTS AND DISCUSSION For the purpose of validating the simulation, experimental data were collected using a laboratory setup which imitates a single air seeder primary conveying line. The apparatus consisted of a straight pipe section connected to a particle introduction point and a fan generated air supply. The dimensions of the test section included an inner pipe diameter of 0.057m and a total length downstream of the particle introduction point of 14m. Wheat was used for the conveyed particles, at a mass flow rate of 2.45 kg/min. The conveying air speed was held constant at 25 m/s at the inlet to the conveying line. These testing conditions are reflective of realistic air seeder operating conditions. Experimental data and validation in the present study was limited to steady-state conditions of fully developed flow. Relevant information for the simulation and testing conditions are summarized in Appendix A.

In a pneumatic conveying system static pressure is both an important and easily measured property of the flow, and thus a convenient means of model validation. Figure 1, presents simulated and experimental pressure data for the described testing conditions. The graphed data shows relatively good agreement between the simulated and experimental results. A very similar shape between the two data sets is evident, with predicted results showing slightly higher overall pressures compared to experiment. Percent error relative to the measured data point, also given in Figure 1, was found to be below 5.0% for 20 of the 22 data points. The average percent error over all data points was calculated at 3.6%, with a maximum value of 14.1%. In terms of absolute error, the average deviation between simulation and experiment was 35.4 Pa, with a maximum value of 66.1 Pa. Closer observation of the pressure data also showed that the greatest absolute deviation between simulation and experiment occurred within the acceleration region following particle induction. This observation is not surprising, given that the drag force model (equation 7) is intended for fully developed flow. From the presented pressure validation data, it can be concluded that the ability of the model and simulation to predict steady-state pressure is relatively good. However, further investigation into a more appropriate drag model for the acceleration zone may improve accuracy further.

Particle velocity was measured experimentally as a second source of data for validation. Measurement of particle velocity has been made using electrostatic sensors recently developed at the University of Saskatchewan. The measured particle velocity data – which represents an average bulk velocity for particles in the axial direction – is shown alongside simulated data in

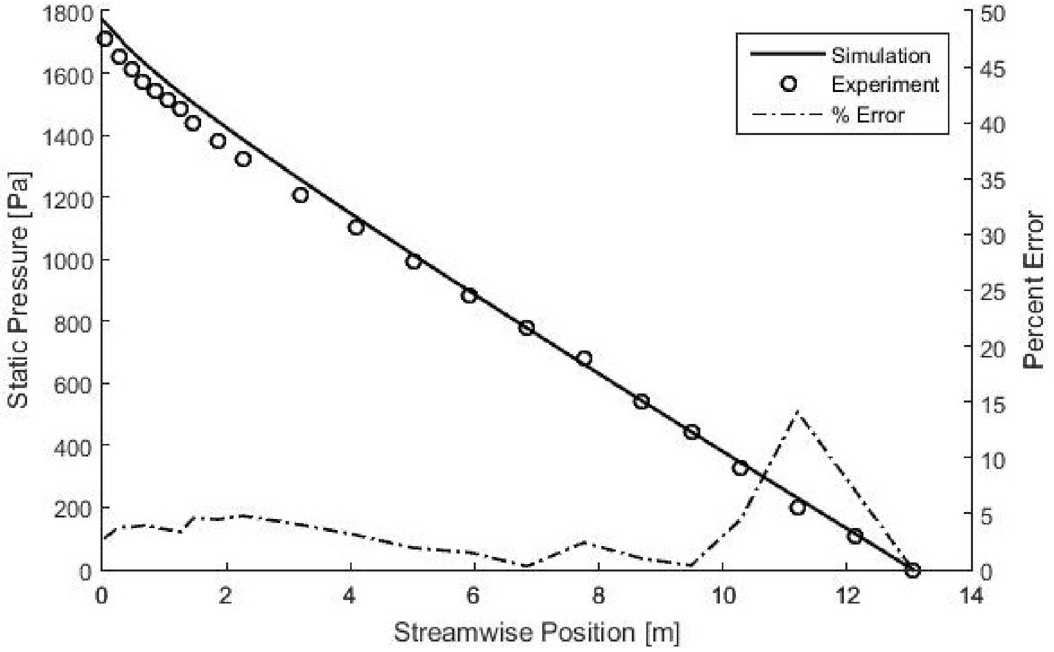


Figure 1: Simulated vs. Experimental Fluid Pressure

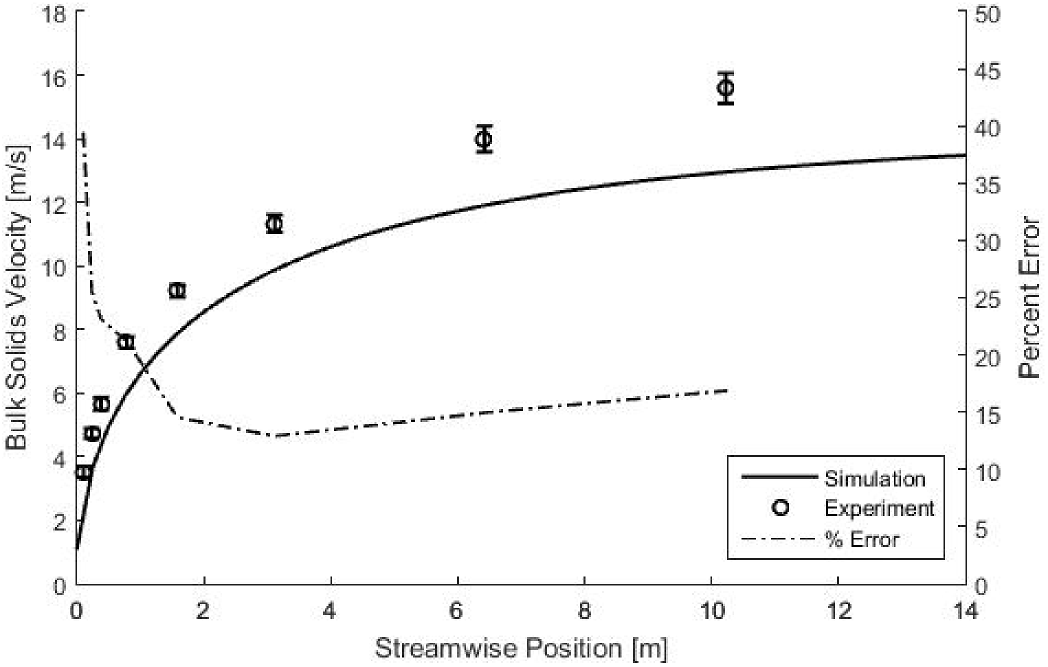
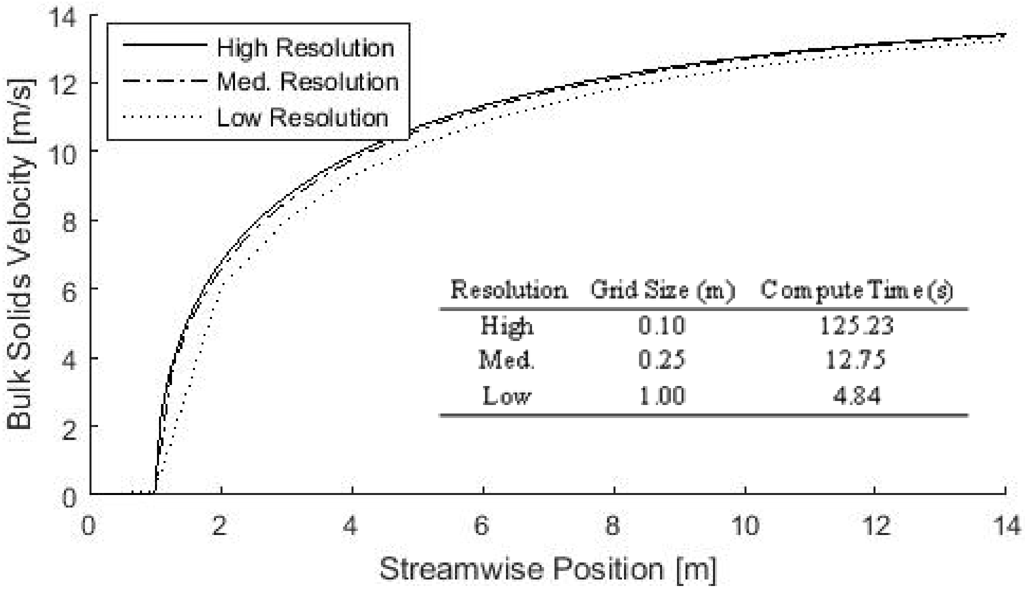


Figure 2: Simulated vs. Experimental Solids Velocity

Figure 2. From observation, it can be seen that the simulated data provided a relatively poor prediction of the experimentally measured solids velocity. Despite exhibiting a very similar profile, the simulated solids velocity is significantly lower than experimental measurements over the entire pipe length. Percent error for the solids velocity can be seen (Figure 2) to vary between 12.9 and 39.3%, with and average value of 18.7%.Absolute error between measured and simulated solids velocity was found to have a maximum value of 2.63 m/s, occurring at the furthest downstream position, and an average value of 1.45 m/s over the entire pipe length. There are several aspects of the model and simulation that could contribute significantly to erroneous velocity prediction; the drag force estimation and the solids velocity boundary condition being the most obvious. For example, applying an offset of +1.40 m/s to the simulated velocity data produces a substantially better fit to the experimental measurements, reducing the average percent error to 2.5% (max 7.9%). This demonstrates, somewhat indirectly, the impact of the solids velocity boundary condition on simulated data, a value which is not known and must be guessed. Therefore, prediction accuracy for the solids velocity may be improved substantially through further investigation and development.

To be useful for control application, simulated results need to have reasonable accuracy while being delivered at a useable speed. Computing time is therefore an important criteria for determining the suitability of the model and simulation. However, computing time is often not a static measurement. It can vary widely depending on factors such as: the conditions being simulated, simulation parameters like resolution of the computational grid or mesh, and the computer hardware being used.

Simulation parameters are often the most significant, and also more easily manipulated by the operator. To examine this aspect of the simulation's performance, several simulations were executed using the same conditions, varying only the resolution of the spatial grid. Results for solids velocity from these three simulations are shown in Figure 3. The focus of these time dependent simulations was a step input of particles from 0 to 2.45 kg/min solids mass flow rate. The remaining parameters were the same as the previously described steady-state conditions, and



Resolution	Grid Size (m)	Compute Time (s)
High	0.10	125.23
Med.	0.25	12.75
Low	1.00	4.84

Figure 3: Simulated Solids Velocity and Computing Time for Different Spatial Resolution

the simulation window was 5.0 seconds. The three simulations included spatial grid increments (i.e. control volume lengths) of 0.1 m, 0.25 m and 1.0 m. The given data shows that although the solution was coarser for lower grid resolutions, as would be expected, the accuracy of the solution changes very little between the three cases. Computing time on the other hand changed substantially, from 125 sec to less than 5 sec between the high and low resolution simulations. Furthermore, the lower resolution seemingly provides similar accuracy to the more expensive simulations, but at speeds approaching real-time. Although this is not a thorough analysis – computing time is for example not evenly distributed across the simulation window – it does provide some optimism that the simulated information could be successfully incorporated into control applications.

CONCLUSION The purpose of the present research was to develop a model and simulation for prediction of multiphase flow conditions found on air seeders, that is suitable for control application. A mathematical model for representing the air-particle flow was selected, and based on the Eulerian-Eulerian or two-fluid modeling framework. A one-dimensional representation of the flow was used, and the model was further simplified where possible; both to increase computational efficiency. A suitable numerical strategy and computer program were developed for solution of the equations. The numerical strategy uses a robust, semi-implicit approach based on the SIMPLER technique. The simulation program was implemented in MatLab. Experimental data were collected with a laboratory apparatus and used to evaluate simulation results. Testing conditions for both experiment and simulation were chosen to be reflective of normal air seeder operating conditions.

Experimental data collected for validation included fluid pressure and solids velocity measurements. Simulated fluid pressure was found to have good agreement with measured pressure data. The average percent error (relative to the measured point) was found to be 3.6% over all data points, with a maximum value of 14.1%. In contrast, simulated solids velocity was found to have poor agreement with experimental data. Although the proper trend was displayed, simulated solids velocity was found to under predict measurements significantly. The average percent error for solids velocity was found to be 18.7% over all data points, with a maximum value of 39.3%. However, there are several components of the model, such as the drag force estimation, which are a likely cause of discrepancy seen in both the pressure and velocity data. Further investigation and improvement of select components of the model may improve the accuracy of simulated data, as well as allow for expansion to a larger variety of physical systems and conditions.

Computing time was examined to evaluate suitability for real-time control application. Overall computing time for the same simulation conditions was found to reduce substantially (from 125 to 5 seconds) when the computational grid was made coarser. The accuracy of simulated data was found to change by a relatively small amount as the grid was changed. Although not exhaustive, this analysis shows potential for the developed model and simulation to be used in control applications.

Acknowledgements The authors would like to thank CNH Industrial for assisting this research; and the previous students of the research program whom helped develop and construct much of the laboratory testing equipment.

NOMENCLATURE

A	cross-sectional area
C_D	drag coefficient
d	diameter
d_h	hydraulic diameter
e	coefficient of restitution for particle collisions
f	friction factor
P	pressure
Re	Reynold’s number
S	source term
t	time
u	mean streamwise velocity
x	axial spatial coordinate
α	void ratio
β	friction coefficient (drag force)
ε	roughness factor
μ	fluid dynamic viscosity
μ_s	solids shear viscosity
ρ	density
θ	granular temperature
$()g$	belonging to gas/fluid phase
$()p$	belonging to particle
$()s$	belonging to solids phase

REFERENCES

Andrews, M.J., and P.J. O’Rourke. 1996. “The Multiphase Particle-in-Cell (MP-PIC) Method for Dense Particulate Flows.” *Int. J. Multiphase Flow* 22 (2): 379–402.

Eskin, D, Y Leonenko, and O Vinogradov. 2007. “An Engineering Model of Dilute Polydisperse Pneumatic Conveying.” *Chemical Engineering & Processing* 46 (3): 247–56. doi:10.1016/j.cep.2006.06.005.

Gidaspow, Dimitri. 1994. *Multiphase Flow and Fluidization : Continuum and Kinetic Theory Descriptions with Applications*. Boston: Boston : Academic Press.

Laín, S., and M. Sommerfeld. 2012. "Characterisation of Pneumatic Conveying Systems Using the Euler/Lagrange Approach." *Powder Technology* 235. Elsevier B.V.: 764–82. doi:10.1016/j.powtec.2012.11.029.

Patankar, Suhas. 1980. *Numerical Heat Transfer and Fluid Flow*. *Series in Coputational Methods in Mechanics and Thermal Sciences*. Washington: Hemisphere Publishing Corporation. doi:10.1016/j.watres.2009.11.010.

Sarrami Foroushani, A, and M Nasr Esfahany. 2015. "CFD Simulation of Gas-Solid Two-Phase Flow in Pneumatic Conveying of Wheat." *Iranian Journal Of Chemistry & Chemical Engineering-International English E* 34 (4): 123–40.

APPENDIX A

Simulation Parameters

pipe inner diameter	d_h	0.05735	m
pipe length	-	14	m
air density	ρ_g	1.1186	kg/m^3
fluid dynamic viscosity	μ	$1.8266 \cdot 10^{-5}$	kg/m·s
pipe roughness	ε	$5.0 \cdot 10^{-6}$	
equivalent particle diameter	d_p	0.00366	m
particle density	ρ_s	1424	kg/m^3
restitution coefficient	e	0.94	
time step size	Δt	0.01	sec
spatial step size	Δx	0.25	m