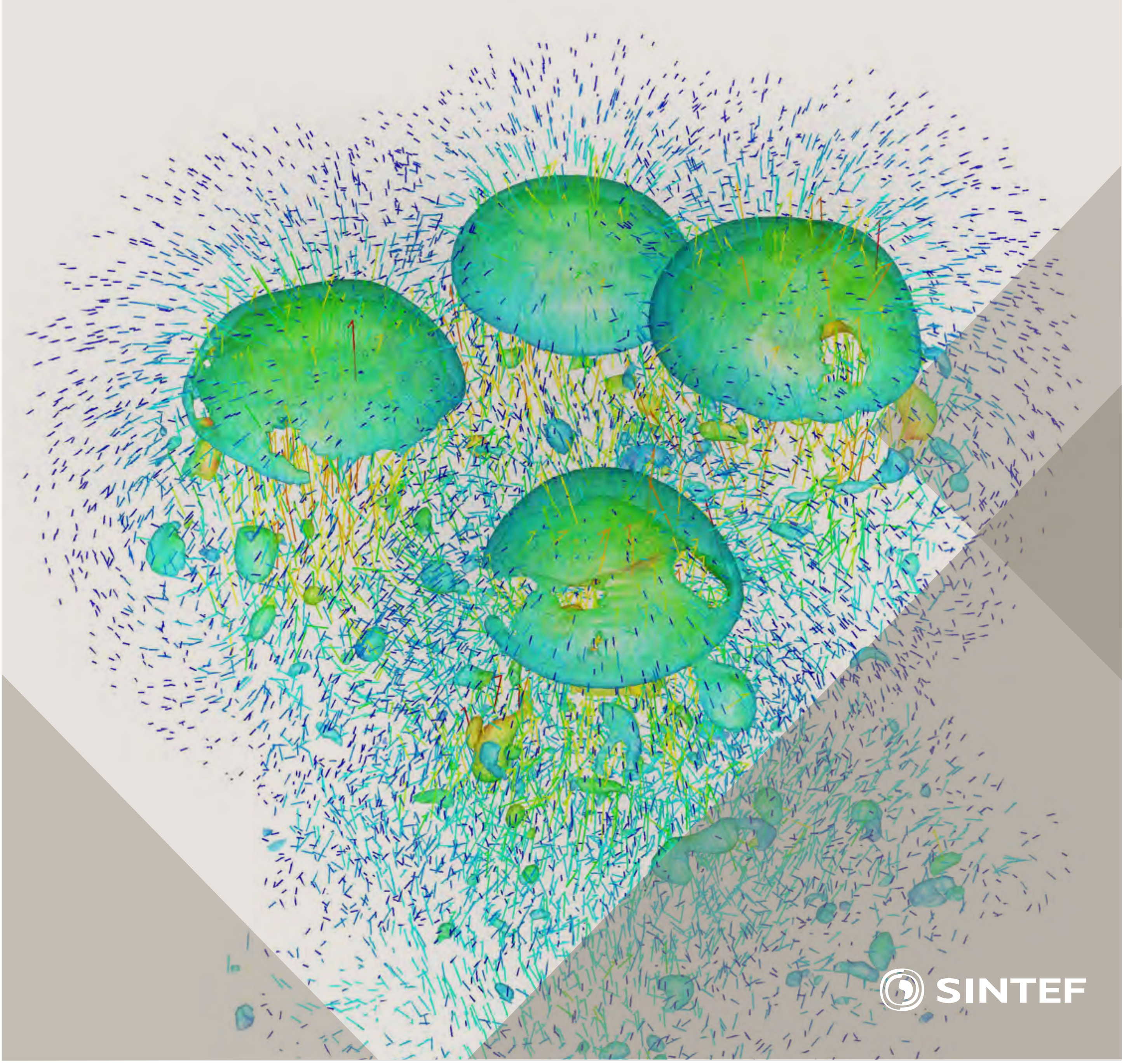
Selected papers from 10th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries

SINTEF
PROCEEDINGS
1

Progress in Applied CFD



SINTEF

SINTEF Proceedings

Editors:
Jan Erik Olsen and Stein Tore Johansen

Progress in Applied CFD

Selected papers from 10th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries

SINTEF Academic Press

SINTEF Proceedings no 1
Editors: Jan Erik Olsen and Stein Tore Johansen
Progress in Applied CFD
Selected papers from 10th International Conference on Computational Fluid
Dynamics in the Oil & Gas, Metallurgical and Process Industries

Key words:
CFD, Flow, Modelling

Cover, illustration: Rising bubbles by Schalk Cloete

ISSN 2387-4287 (printed)
ISSN 2387-4295 (online)
ISBN 978-82-536-1432-8 (printed)
ISBN 978-82-536-1433-5 (pdf)

60 copies printed by AIT AS e-dit
Content: 100 g munken polar
Cover: 240 g trucard

SINTEF Academic Press
Address: Forskningsveien 3 B
PO Box 124 Blindern
N-0314 OSLO
Tel: +47 22 96 55 55
Fax: +47 22 96 55 08

www.sintef.no/byggforsk
www.sintefbok.no

SINTEF Proceedings
SINTEF Proceedings is a serial publication for peer-reviewed conference proceedings on a variety of scientific topics.
The processes of peer-reviewing of papers published in SINTEF Proceedings are administered by the conference organizers and proceedings editors. Detailed procedures will vary according to custom and practice in each scientific community.

PREFACE

This book contains selected papers from the 10th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries. The conference was hosted by SINTEF in Trondheim in June 2014 and is also known as CFD2014 for short. The conference series was initiated by CSIRO and Phil Schwarz in 1997. So far the conference has been alternating between CSIRO in Melbourne and SINTEF in Trondheim. The conferences focus on the application of CFD in the oil and gas industries, metal production, mineral processing, power generation, chemicals and other process industries. The papers in the conference proceedings and this book demonstrate the current progress in applied CFD.

The conference papers undergo a review process involving two experts. Only papers accepted by the reviewers are presented in the conference proceedings. More than 100 papers were presented at the conference. Of these papers, 27 were chosen for this book and reviewed once more before being approved. These are well received papers fitting the scope of the book which has a slightly more focused scope than the conference. As many other good papers were presented at the conference, the interested reader is also encouraged to study the proceedings of the conference.

The organizing committee would like to thank everyone who has helped with paper review, those who promoted the conference and all authors who have submitted scientific contributions. We are also grateful for the support from the conference sponsors: FACE (the multiphase flow assurance centre), Total, ANSYS, CD-Adapco, Ascomp, Statoil and Elkem.

Stein Tore Johansen & Jan Erik Olsen

Organizing committee:

Conference chairman: Prof. Stein Tore Johansen
Conference coordinator: Dr. Jan Erik Olsen
Dr. Kristian Etienne Einarsrud
Dr. Shahriar Amini
Dr. Ernst Meese
Dr. Paal Skjetne
Dr. Martin Larsson
Dr. Peter Witt, CSIRO

Scientific committee:

J.A.M.Kuipers, TU Eindhoven
Olivier Simonin, IMFT/INP Toulouse
Akio Tomiyama, Kobe University
Sanjoy Banerjee, City College of New York
Phil Schwarz, CSIRO
Harald Laux, Osram
Josip Zoric, SINTEF
Jos Derksen, University of Aberdeen
Dieter Bothe, TU Darmstadt
Dmitry Eskin, Schlumberger
Djamel Lakehal, ASCOMP
Pär Jonsson, KTH
Ruben Shulkes, Statoil
Chris Thompson, Cranfield University
Jinghai Li, Chinese Academy of Science
Stefan Pirker, Johannes Kepler Univ.
Bernhard Müller, NTNU
Stein Tore Johansen, SINTEF
Markus Braun, ANSYS

CONTENTS

10th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries
SINTEF, Trondheim, Norway *CFD 2014*
17-19 June 2014

MULTI-SCALE PROCESS MODELS TO ENABLE THE EMBEDDING OF CFD DERIVED FUNCTIONS: CURTAIN DRAG IN FLIGHTED ROTARY DRYERS

Andrew LEE[2], Madoc SHEEHAN[1] , Philip SCHNEIDER[1]
[1] James Cook University, 4812 Townsville, AUSTRALIA
[2] University of Melbourne, 3010 Melbourne, AUSTRALIA

Email: madoc.sheehan@jcu.edu.au

ABSTRACT

Flighted rotary dryers are large industrial devices which are commonly used to dry mineral ores and mineral concentrates, as well as other valuable commodity products. They are high capital cost units as well as large consumers of energy. Solids movement and energy exchanges within these devices occurs via a range of complex mechanisms that involve rolling and bouncing in a dense bed of solids, as well as the falling through a cross-flowing gas stream in lean particle curtains. Although a fundamental approach is attractive, full CFD simulations of such devices would be prohibitively expensive. The complexity of such a model would preclude its use for design and control applications, which are the most prevalent concerns to industry. Pseudo-physical compartment modelling is a powerful alternative technique that can be used to reproduce, in a physically meaningful way, the important characteristics of dryers such as residence time distributions and loading states. This scalable modelling approach also provides a convenient multi-scale structure that facilitates the representation of a system (in this case a flighted rotary dryer) as a series of smaller, distinctive, interacting phases. It is these smaller phase structures, such as the air-borne phase, that are suitable for modelling with either CFD or DEM type approaches. In this paper CFD modelling of single particle curtains and multiple side-by-side particle curtains is presented, with particular emphasis on quantifying gas induced drag and gas penetration into the curtain phase. The results are discussed in terms of their suitability to integrate CFD derived phase information within the broad process model. The simulations described in this paper provide valuable insights into the dryer design considerations such as flight serrations and axial flight staggering. The methodology presented in this paper provides an example that could be adapted to enable the evaporation, convection and radiation heat transfer in curtains to be accounted for.

Keywords: CFD, compartment model, particle curtain, drag, multi-scale, dryers

INTRODUCTION

Flighted rotary dryers (FRD's) are used extensively in a range of industries for control of the temperature and moisture content of free flowing, particulate solids, such as grains, sugar, and mineral ores as shown in cross-section, in Fig.1. FRDs range from small bench scale apparatus in pharmaceutical manufacturing, to 30m long, 6m diameter, industrial ore dryers. FRDs offer simplicity, low operating costs, and handle a wide range of throughputs and feed-stocks. Due to their size, rotary dryers often represent a significant capital expense. Thus it is necessary to have a good understanding of dryer operations and design features to ensure that the unit meets desired operational requirements.

Many different types of flighted rotary dryers exist, including multi-pass units and units with centre fills. However the simplest and most common flighted rotary dryers consist of a rotating, inclined drum with lifters or flights fitted to the internal walls. Moist solids are fed into the dryer at one end where they are collected in the flights. The flights carry the solids into the upper half of the drum, where they are released in a series of continuous curtains across the width of the dryer (see Figure 1). These particles fall under the influence of gravity and return to the floor of the dryer where they are collected once again by the flights. Axial transport of solids within the dryer is caused by the slope of the drum, and occurs via both the cascading of solids off flights and rolling/kilning motion of solids along the drum base. Drying gasses, commonly air or combustion gasses, are fed through the dryer either co- or counter-currently. These gases interact with the falling curtains of solids, removing heat and moisture and creating drag forces that will influence the curtains of falling particles, causing dispersion of the solids within the dryer. The rolling solids also interact with the drying gasses, but to a lesser extent.

Whilst flighted rotary dryers are widely used and have been widely studied, their complex solids transport behaviour and the difficulty in integrating solids transport and heat and mass transfer phenomena have proved to be significant stumbling blocks to modelling. As a rotary dryer and the behaviour of its contents are three-dimensional and vary with time, a comprehensive first principles dynamic model of a dryer would need to be multi-dimensional in order to capture the full detail of the solids transport. Furthermore, both granular flow (flight-borne and kilning solids) and pneumatic flow (curtaining solids) occur within these devices. Whilst a combined CFD and DEM model can be conceptualised, the particle numbers and mesh nodes necessary to model a full scale unit would be enormous. Even ignoring the uncertainties in first principles modelling of granular flow, the numerical demands of such a model would preclude its practical use for either design or control purposes. Clearly, a rigorous fluid dynamic or discrete element model of an entire dryer is not feasible, and a more pragmatic modelling approach is necessary for guiding the development of industry-scale process models. In what follows we have been guided by the principles of pragmatism, attempting to use as simple a model as possible while still capturing important influences on system behaviour.

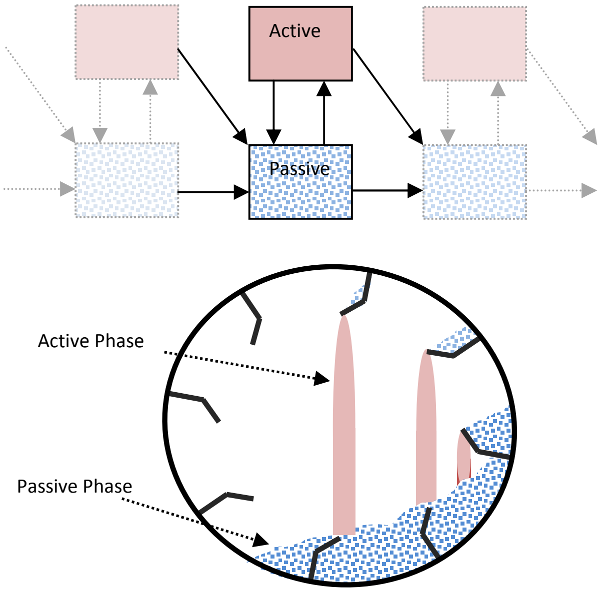


Figure 1. Cross section of an operating flighted rotary dryer (below) and the corresponding pseudo-physical compartment model (above). The light-shaded solids are the airborne solids (curtains) and the patterned solids are the flight and drum borne solids.

Unfortunately simple empirical models have also been unable to predict the full behaviour of the dryer systems (Cao and Langrish (1999), Lee *et al.* (2009)). In particular, the influence of the flights on system performance has proven difficult to capture with empirical techniques. Early semi-empirical applications of compartment modelling of FRD's by both Matchett and Baker (1987), and Duchesne *et al* (1996) led to the development of more realistic representations of solids transport in flighted rotary dryers. The model structures they suggested were capable of reproducing typical dryer experimental observations such as residence time distributions. The important model features developed in these works were the use of a twin tanks in series model structure, where one tank represented the flight and drum borne solids and the other tank represented the airborne solids (see Figure 1). In these models, the exchange rates moderating solids flows between the tanks, such as the flow of airborne solids off the flights and onto the base of the drum, were parameter estimated. Sheehan *et al.* (2005) and Britton *et al.* (2006) extended this work by developing a multi-scale flighted rotary dryer model, which integrated physically-derived (i.e. mechanistic) model parameters, instead of statistically estimated parameters, into the twin-tanks in series compartment model structure. They called this approach pseudo-physical compartment modelling (PPCM). The PPCM is an adaptable pragmatic model framework, and it has been successfully used to model industrial sugar drying (Britton *et al.* (2006), a full-scale zinc concentrate dryer with both flighted and unflighted sections (Ajayi, 2011) and a fluidised drum granulator (Rojas *et al.*, 2010). In the PPCM approach, scalable dynamic models are derived which are capable of reproducing experimentally observable features (such as the residence time distribution and holdup profiles). Furthermore, the effects of internal geometry, operating conditions and solids flow properties on dryer performance could also be accurately predicted and integrated into the model equations in a meaningful way.

Similar to the network of zones approach in CFD, each well-mixed tank in the PPCM has a distinct physical representation. Each tank is defined by its corresponding dynamic mass and energy conservation equations, as well as geometrically defined capacity constraints. For example, an active phase tank in the model physically represents (within a distinct slice of the dryer) the curtains of particles that cascade off the flight tips as the drum rotates. The corresponding passive phase tank physically represents the solids in the flights and on the floor of the drum. Capacity constraints, exchange rates between these tanks and between adjacent tanks are described using mechanistic arguments, based solely on solids flow properties and drum and flight geometry. A separate unloading flight geometry model was used to generate the model parameters controlling the exchange rates in and out of each tank or phase.

Considering the exchange rate characterising the flow between the passive and active tanks (i.e. flight discharge), Lee and Sheehan (2010) showed that for a free flowing material which discharges continuously from the flights, the discharging behaviour of a flight as it travels around the circumference of the dryer can be calculated solely from the geometry of the flight and drum, the amount of solids present in the flight, and the solids dynamic angle of repose. Thus, an unloading profile of a flight (mass flow versus time) can be calculated as a function of the drum loading.

In the discussion that follows, we describe aspects of the PPCM process model structure that help to outline the potential to utilise CFD outputs to regress model parameters within the PPCM. In particular, we emphasise the model parameters that moderate active phase flows (i.e. the curtains of particles that cascade off the flights) that are currently predicted using the geometry model, but could instead be generated using CFD simulations. It important to note that because of the scaling associated with the compartmentalisation of the full scale unit, these simulations would be of a more tractable size and reduced level of complexity compared to a first principles simulation.

Two key parameters utilised in the PPCM are the airborne solids residence time, and the horizontal drag experienced by the curtains due to cross-flowing gas. For a specific example, the solids flow rates out of each compartment in the model are calculated using the relation $F_i = m_i / \overline{t}_i$ where F_i is the flow rate of solids leaving compartment i and m_i and $\overline{t_i}$ are the solids holdup and average residence time in compartment i, respectively. Currently, to evaluate drag and residence time correlations used within the PPCM, the unloading flight geometry sub-model is used as boundary conditions for solving Newtons equations of motion for a isolated particle. The bulk curtain behaviour is estimated by calculating the path or trajectory of a single spherical falling particle, including displacement due to cross flowing gas induced drag. The critical assumption inherent in this approach is that the bulk curtain behaviour can be assumed to be adequately represented by the motion of an isolated sphere. Furthermore, heat transfer in the curtains is also approximated by assuming correlations (such as the Ranz Marshall correlation) based on an assembly of isolated spheres.

Unfortunately there is strong evidence that curtains of particles behave significantly differently to isolated spheres (Wardjamin *et al*. (2008, 2009) and Hruby *et al*. (1988)). Because of gas entrainment, curtains exhibit reduced residence times and reduced drag in comparison to isolated spheres. Although simplifying the modelling, the assumption that an isolated sphere adequately represents the airborne solids can be improved upon, through the use of CFD. For example, CFD can be used as a tool to progressively improve the prediction of drag and residence time (and also heat transfer) by modelling either a single (i.e. average) particle curtain or for an increased level of sophistication, modelling a series of particle curtains corresponding to the discharging flights. It is worth noting here that the potential to utilise CFD sub-modelling to improve overall model predictions is only made possible by the enforced physical realism inherent in the pragmatic compartment-model structure. In this way the PPCM methodology facilitates varying degrees of pragmatism to be used to incrementally and systematically improve representation of the underlying phenomena.

In this paper we use the case study of a flighted rotary dryer as context for describing new CFD modelling of drag within particle curtains. We describe the CFD modelling of a single cascading curtain as well as multiple cascading curtains and also discuss the potential for embedding CFD derived information within multi-scale process compartment models.

CFD curtain modelling

A number of researchers have studied the effects of the gas-solids interactions in the active phase on the solids transport operations within a rotary dryer. Obviously, the moving stream of drying gasses will result in a displacement of the falling solids due to drag effects. Previous research has demonstrated that the gas-solids interactions within a rotary dryer are complicated. Simplifications of the system, such as the extremes of assuming isolated spherical particles (maximum drag) or flat-plate behaviour (minimum drag) (Baker, 1992), are generally insufficient. However, modelling the gas-solids interactions within a single curtain of falling solids is well within the capabilities of current CFD packages.

In order to model the gas-solids interactions influencing the active phase, a multi-phase CFD approach was implemented in Ansys CFX$^{®}$ 5.7.1. Due to the large number of solid particles present a Lagrangian method was not feasible, whilst the solid curtain was not considered dense enough to warrant the use of a granular model for the solids. Thus an Eulerian-Eulerian approach was used with a full buoyancy model and a k-ε model for turbulence. Although the k-ε model remains the industry standard, and has been widely used to model gas-solid systems (Du *et al*. (2006) for example), it's accuracy has been noted to depend on the use of sufficiently fine mesh and time step (Fletcher *et al*. (2006)). Uncertainty in the choice of turbulence modelling emphasizes the importance of experimental data to validate model predictions. In this work we draw confidence in our model choices and the resulting simulation results, from prior studies comparing experimental (isothermal) curtain profiles to the simulated curtain profiles that were determined using the same model approach, but for a scaled-down system. In those wind tunnel studies (Wardjiman *et al.* (2008, 2009)), both free falling curtains in still air and free falling curtains exposed to cross flowing gas were examined. Good agreement between the experimental and simulated results was obtained under both scenarios. Wardjiman *et al.* (2008, 2009) found that a cutoff solids volume fraction of $5.6\text{x}10^{-4}$ corresponded well with the experimentally observed curtain boundaries, and a similar threshold of $4.3\text{x}10^{-4}$ has been used in this work. However, to ensure complete confidence in the results presented in this paper, a comprehensive set of experimental data for curtain profiles would be required.

Gas-solids interactions occurring within a full rotary dryer are complex, with multiple flights unloading solids at different locations. Each of these individual curtains will have different mass flow rates, particle velocities and solids volume fractions, and the presence of the curtains will affect the flow of gas through the dryer. In this work, two series of simulations were conducted. The first studied a single curtain of solids falling perpendicular to a moving gas stream (i.e. a single curtain in isolation) and the second examined multiple parallel curtains to determine whether the presence of multiple curtains affected the displacement of solids and extent of gas penetration.

Single Curtain Studies

In order to achieve meaningful results from the single curtain simulations, it was necessary to determine a set of initial conditions that would represent the average gas-solids interactions for the falling curtain in a typical industrial dryer. The curtain conditions for the average fall path of a particle were used, based on the predictions of the geometric unloading model of Lee and Sheehan (2010) and experimental observations (Lee, 2008). These gave an initial curtain width of 18 mm with an initial vertical particle velocity of 1 m/s and a mass flow rate per metre of flight length of 5.18 kg/m·s. The single curtain studies were simulated in a tunnel 0.8 m long, 0.52 m wide and 2 m tall. Solids were introduced through an 18 mm wide variable length inlet along the centreline of the tunnel, allowing a 0.1m entry zone for the gas stream. Turbulence was modelled using the k-ε model and a full buoyancy model was used. Drag forces were modelled using the Schiller-Naumann (Schiller and Naumann, 1935) equation assuming a particle size of 850 µm.

The numerical domain was discretised with a 9 mm tetrahedral mesh applied to the region occupied by the particle curtain and extended for a minimum of 50 mm beyond the expected curtain boundaries. The remainder of the tunnel was discretised using a maximum mesh size of 35mm, resulting in a total mesh of 136,000 nodes. The system was solved using the inbuilt Automatic Timescale calculator in CFX 5.7.1 (Ansys CFX (2006)) until all residuals were less than 10^{-4}, or 100 iterations had been performed. The gas velocity at the inlet to the duct was specified at the experimental conditions and at the solids inlet, the mass flow rate, velocity and initial solids volume fraction were specified. The inlet solids volume fraction of solids, r_{p0}, was calculated based on experimental data using the following equation $r_{p0} = \dfrac{\dot{M}_p}{\rho_p U_0 A}$, where $\dot{M}_p$ is the mass flow rate of solids entering the system, U_0 is the initial velocity of the solids entering the duct, and A is the cross-sectional area of the solids inlet. At the downstream end of the duct, the boundary was defined as an outlet boundary, such that material can only exit the system through this boundary. In the absence of experimental data, the gas and solids inlets were given a turbulence intensity of 5% (Ansys CFX (2006)). The remaining boundary conditions were governed by the no-slip condition.

In order to test the influence of gas velocity on the curtain displacement, simulations were conducted with a curtain length of 0.5m, and gas velocities of 0.5 m/s (slow), 1 m/s (normal gas velocity for rotary dryers) and 2 m/s (fast). Although not presented here, simulations were also conducted with 5 different curtain lengths (i.e. axial distances) to study how far the moving gas stream penetrated the falling curtain, and how solids displacement varied with curtain length.

Figure 2 shows the simulated results for the curtain profile measured at the centreline of the tunnel, for a 0.5 m long curtain at different gas velocities. It can be clearly seen that the gas velocity has a significant effect on the leading edge of the curtain, with increasing gas velocities causing greater displacement of the solids. At the trailing edge however, the profile is very similar at both 0.5 m/s and 1 m/s, with the solids falling almost vertically under these conditions. It is only with a gas velocity of 2 m/s that the gas is able to fully penetrate and the trailing edge of the curtain is displaced.

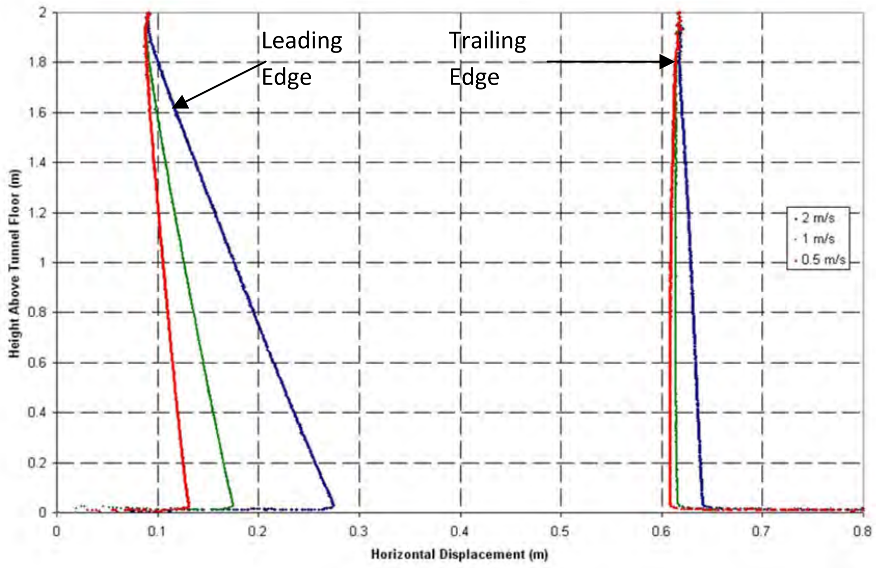


Figure 2. Solids curtain profile at different gas velocities (0.5 - 2m/s).

Figure 3 shows a colour map of the horizontal component of the gas velocity for the 1 m/s simulation. These measurements were taken in the horizontal plane at a height of 1 m above the tunnel floor (arbitrarily chosen to illustrate the behaviour of the system), with the gas being introduced from the bottom of the figure. The black line indicates the edge of the particle curtain (defined as a solids volume fraction of 0.43×10^{-3}). Figure 3 clearly shows the gas being channelled around the solids curtain, with increases of up to 25% in the gas velocity being observed around the curtain. It can also be seen that the moving gas only penetrates a short distance into the solid curtain, producing an area of negligible horizontal velocity throughout large portions of the curtain. Figure 4 shows the same measurement as Figure 3 for an initial gas velocity of 2 m/s, and again gas velocities up to 25% greater than the initial velocity can be observed. Clearly, the increased gas velocity has a significant impact on the leading edge of the curtain, causing appreciable curtain displacement and compression. It can also be seen that the moving gas penetrates deeper into the curtain than in the 1 m/s simulation, however there is still an area of negligible horizontal gas velocity throughout large portions of the curtain. Given that flights are often staggered to maximise gas-solid interactions, these results provide useful guides to appropriate spacing. In industry flights are typically staggered at around 1-2 m intervals, yet this study suggests that staggering of between 10 and 20 cm would be more effective in promoting gas penetration.

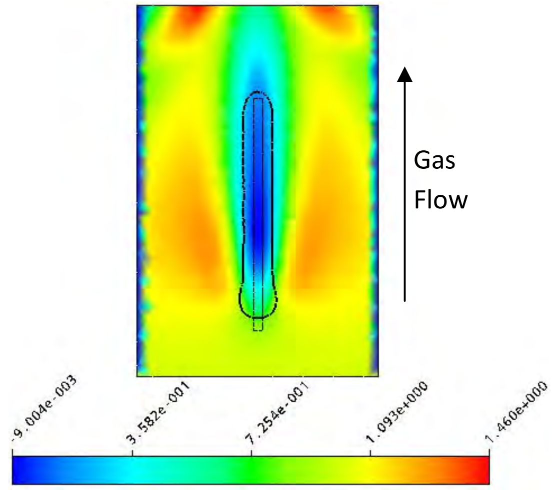


Figure 3. Horizontal gas velocity colour map at 1 m above tunnel floor (1 m/s initial gas velocity, 0.5 m inlet, 5.18 kg/m.s solids flow rate). Units are m/s.

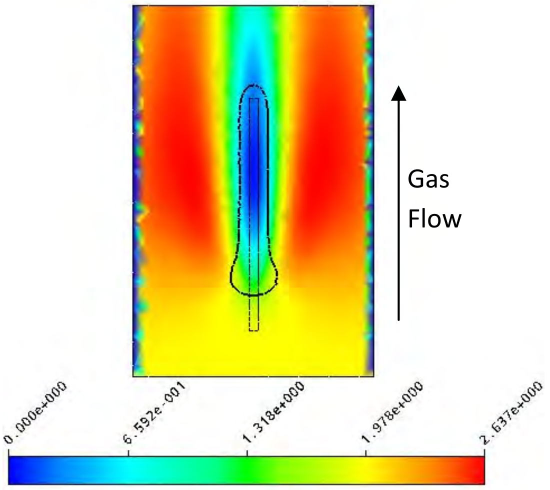


Figure 4. Horizontal gas velocity colour map at 1 m above tunnel floor (2 m/s initial gas velocity, 0.5 m inlet, 5.18 kg/m.s solids flow rate). Units are m/s.

From the simulation results, it is possible to assess curtain displacement resulting from changes in gas velocity. Figure 5 shows the trend in the displacement of the curtains leading edge over a fall of 2 metres at the gas velocities studied. The curtain displacement is well represented by a second order polynomial, which reconciles with fact that drag is proportional to velocity squared.

Simulations such as this provide an example whereby CFD can be used to regress correlations that can be embedded in the PPCM of a industrial flighted rotary dryer. In this case, PPCM parameters (e.g. curtain displacement) become functions of operating variables such as gas velocity. Extensions could include developing CFD derived regressions that also account for the effects of solids flow rates and/or particle size distribution. Furthermore, using averaged CFD curtain properties such as the fall time (i.e. mean residence time in the active phase), can be used to replace isolated sphere model predictions. This would be advantageous because it is well known that drag in curtains is different to that experienced by isolated single particles (Hurby *et al.* (1988)).

The predicted displacement of the solids due to gas-solids interactions appears small in comparison to those reported by Baker (1992), which is likely due to different initial conditions of the curtain. In these simulations, the initial conditions were taken using experimental data from a flight unloading apparatus with an un-serrated flight (Lee and Sheehan, 2010). However, most industrial dryer use serrated flights, which create a broader, less dense curtain. This reduced density allows greater gas-solids interaction and thus greater displacement of solids. Another possible factor affecting the results is the interactions between curtains. In a system with multiple curtains there will not be as much space available for channelling around the curtains, which may cause greater interactions between the gas and solids.

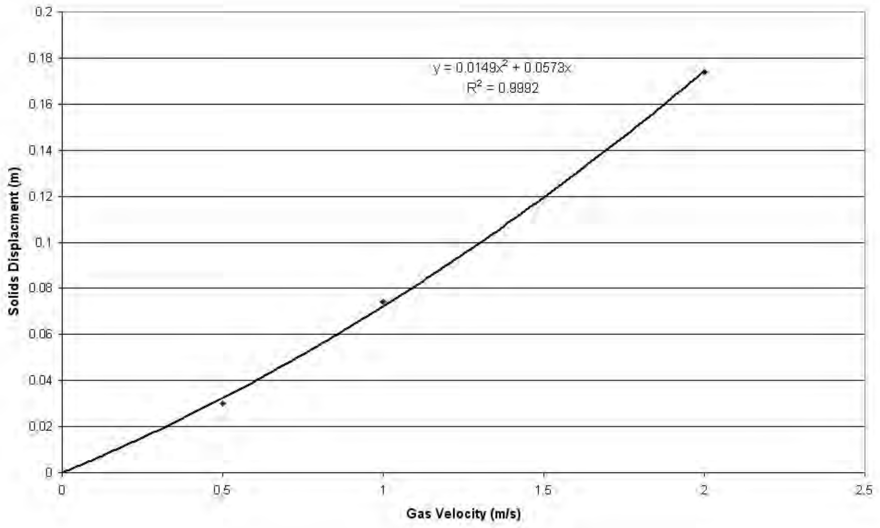


Figure 5. Simulated single curtain displacement over 2 metres at different gas velocities.

Multiple Curtain Studies

The phenomenon of channelling of the gas flow around the falling curtains in a rotary dryer was first considered by Baker (1992) as part of his studies into the gas-solids interactions. In a system with multiple curtains, the gas flow should channel between curtains, resulting in channels of high gas velocity that may influence solid particles at the curtain edges. In order to understand channelling behaviour, a series of simulations were conducted using multiple curtains with different spacing between curtains.

CFD simulations were conducted using the average particle curtain properties used for the single curtain simulations for all curtains, which is a reasonable approximation for the middle of the unloading profile where the unloading rate is reasonably constant. The simulations were conducted using a tunnel 500 mm high, 340 mm across and 800 mm long. A smaller system geometry (compared to the single curtain simulations) was used due to the greater mesh requirements to resolve the gas-solids interactions for the multiple curtains. Solids were introduced along the top of the tunnel starting 100 mm from the gas inlet, and allowing 200 mm between the end of the inlet and the end of the tunnel. The tunnel was discretised with an 8.4 mm tetrahedral mesh across the entire tunnel, resulting in 368,780 nodes. Four simulations were run with curtain spacing varying between 50 mm and 80mm. Curtains were equally spaced across the tunnel, with the two outermost curtains extended to reach the wall to prevent gas for being channelled along the wall without affecting the curtain. Due to the fixed size of the tunnel, the constant curtain spacing meant that the number of curtains present in the simulations also varied with curtain spacing, making extrapolations based on total curtain numbers difficult.

Figures 6 to 7 show the simulated solids volume fraction at a horizontal cross-section 0.25 m above the floor of the tunnel (halfway through the fall) for the different curtain spacing, with gas entering from the right hand side at 1m/s. The black boxes show the location of the solid inlets. As can be seen, with a curtain spacing of 50 mm, the curtains have merged to form a single broad curtain, but as the curtain spacing increases to 80mm, individual curtains begin to become apparent. This agrees well with the results reported by Wardjiman *et al.* (2009). In these experiments, it was observed that falling curtains of solids with high solids volume fractions tended to expand until a stable state was reached. They proposed that this was due to the difference in pressure inside the curtain compared to outside, causing the curtain to expand until the pressures equalised. This explains the expansion of the curtains observed in the multiple curtain simulations, resulting in the merging of the curtains into a uniform phase when the curtain spacing is small enough.

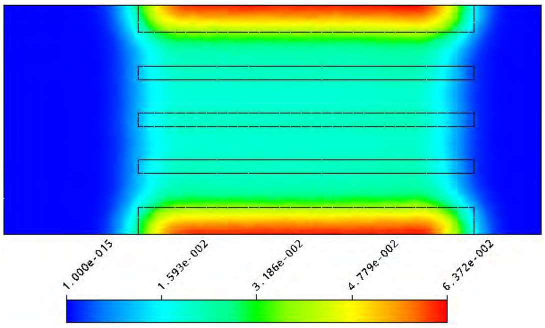


Figure 6. Solids volume fraction colour map at 0.25 m above tunnel floor with 50 mm curtain spacing.

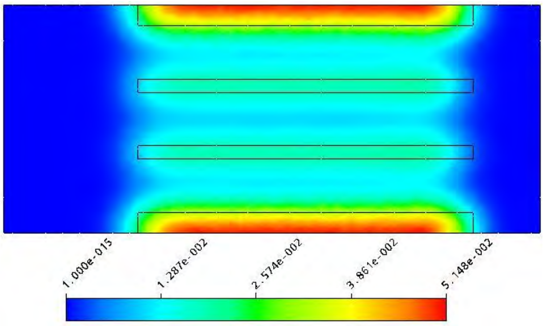


Figure 7. Solids volume fraction colour map at 0.25 m above tunnel floor with 80 mm curtain spacing.

This means that at low curtain spacing, where the curtains have merged into a single uniform phase, the gas flow through the curtain should also be uniform, as there are no regions of reduced solids volume fraction through which the gas will channel. As the curtain spacing increases, and individual curtains become distinct, and the regions of lower solids volume fraction between the curtains will allow for the gas to be channelled. This effect is seen in Figures 8 and 9, which show the gas velocity profile through the same cross-section of the tunnel. The thick black line indicates the contour of a solids volume fraction of 4.3×10^{-3}.

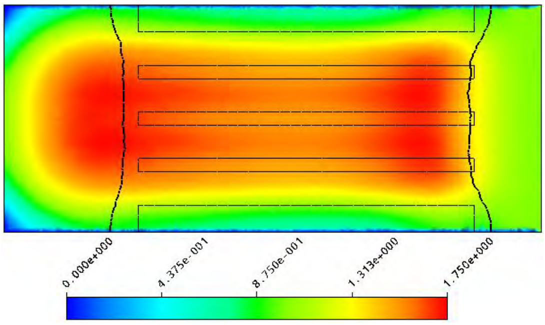


Figure 8. Horizontal gas velocity colour map at 0.25 m above tunnel floor with 50 mm curtain spacing. Units are m/s and the inlet gas velocity is 1m/s.

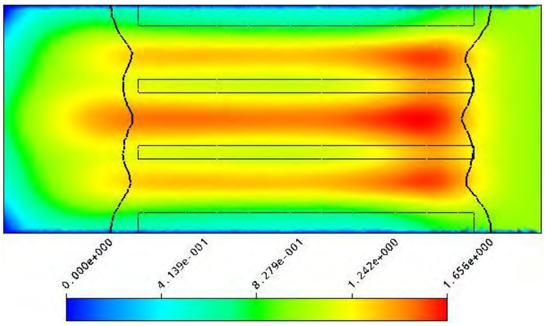


Figure 9. Horizontal gas velocity colour map at 0.25 m above tunnel floor with 80 mm curtain spacing. Units are m/s and the inlet gas velocity is 1m/s.

As can be seen, the channelling of gas flow between the curtains increases with increasing curtain spacing. At a curtain spacing of 50 mm and a fall height of 0.25 m, the gas velocity through the combined curtains is close to uniform, whilst at a curtain spacing of 80 mm significant channelling of the gas flow can be seen. Larger velocity differences between the cross-flowing gas and falling solids within the curtains would lead to enhanced heat and mass transfer. Clearly CFD simulations involving heat and/or mass transfer would be an important step defining these potential enhancements, and would have significant repercussions for flight design, and selecting the appropriate number of flights/curtains. In these simulations, it can be seen that there is significant gas velocity within the curtains, unlike the single curtain experiments where the gas velocity within the curtain could be considered negligible in comparison. This increased gas velocity within the falling curtain of solids would result in an increased displacement of the solids compared to the single curtain simulations, as was shown in Figure 10. Figure 10 shows the profile of the falling curtain, defined by a solids volume fraction of $4.3\text{x}10^{-3}$, for the different simulations compared to the single curtain simulation with the same initial conditions. The curtain profile was measured along the centreline of one of the solid inlets.

It can be clearly seen that the single curtain simulation predicts less displacement of the solids curtain, due to the channelling of the gas around the curtain. It can be seen that the effects of the gas-solids interactions occur primarily at the leading edge of the curtain, with a significant displacement of solids being observed.

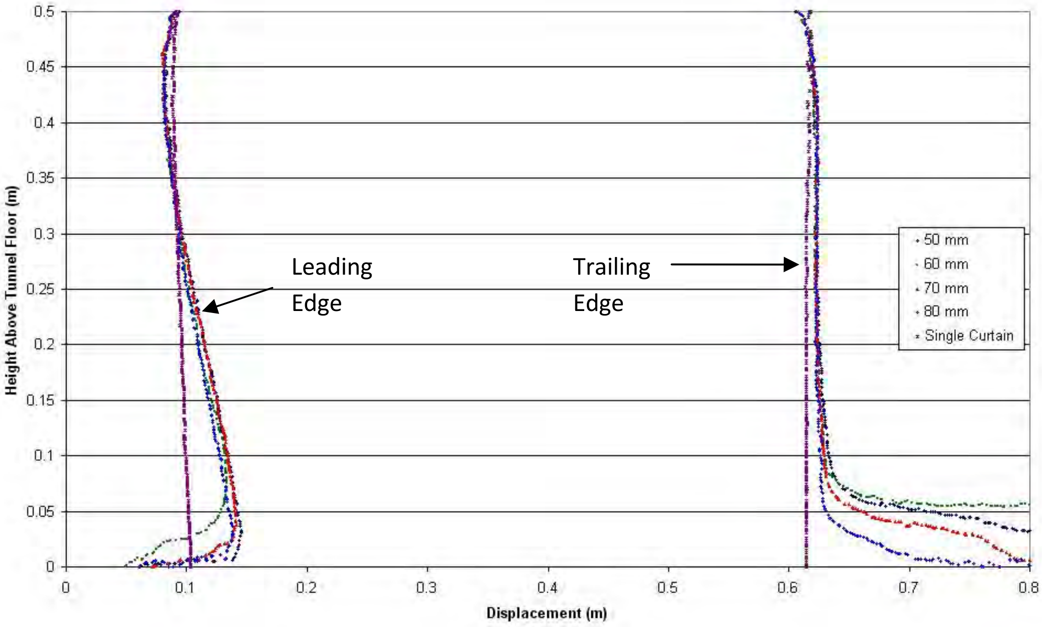


Figure 10. Curtain profiles for different curtain spacing and cross flowing gas at 1m/s.

CONCLUSIONS

The pseudo-physical compartment model (PPCM) is presented as a convenient structure to reduce the size and scale of modelled phenomena whist maintaining physical realism. In this case, a flighted rotary dryer is modelled in such a way that the behaviour of falling curtains of particles can be compartmentalised and emphasised. Eularian-Eularian CFD simulations of gas induced drag on the curtain phase within a flighted rotary dryer are described. Both single curtain simulations and multiple curtain simulations are presented. Examples illustrating the use of CFD results to develop correlations suitable for use within the PPCM are described.

Single curtain simulations show substantial channelling of gas around the sides of the particle curtain, leading to reduced particle drag (except at the leading edge) and low levels of gas penetration (20cm or less at 2m/s gas velocity). However, simulation of multiple curtains lead to more significant gas penetration that is dependent on the curtain to curtain separation. In terms of particle drag, the displacements of solids in the single curtain simulations with 850 µm particles are less than expected. The simulations conducted in this study used data measured from unserrated flights, resulting in thinner, denser curtains, that potentially under-predict solids displacement that would be observed when using serrated flights. An experimental study of the unloading behaviour of serrated flights is necessary in order to develop a better model for the gas-solids interactions in flighted rotary dryers.

REFERENCES

AJAYI OO (2011) Multiscale modelling of industrial flighted rotary dryers. PhD thesis, School of Engineering and Physical Sciences, James Cook University, Townsville.

Ansys CFX Solver Modelling Guide, (2006), Ansys Inc., Canonsburg

BAKER, C.G.J., (1992), “Air-Solids Drag in Cascading Rotary Dryers”, *Drying Technology*, **10**, 365-393

BRITTON P.F., SHEEHAN M.E. and SCHNEIDER P.A., (2006) A physical description of solids transport in flighted rotary dryers. *Powder Technology* **165**, 153-160.

CAO, W.F. and LANGRISH, T.A.G., (1999) Comparison of residence time models for cascading rotary dryers. *Drying Technology*, **17** (4&5), 825-836.

DU, W., BAO, X., XU, J. and WEI, W., (2006) “Computational fluid dynamics modelling of spouted bed: assessment of drag coefficient correlations”, *Chem. Eng. Science*, **61**, 1401-1420

DUCHESNE, C., THIBAULT, J. and BAZIN, C., (1996) “Modelling of the Solids Transportation within an Industrial Rotary Dryer: A Simple Model”, *Ind. Eng. Chem. Research*, **35**, 2334-2341

FLETCHER, D.F., GUO, B., HARVIE, D., LANGRISH, T.A.G., NIJDAM, J.J. and WILLIAMS, J., (2006) “What is important in the simulation of spray dryer performance and how do current CFD models perform? ”, *App. Math. Modelling*, **30**, 1281-1292

HURBY, J., STEEPER, R. EVANS, G. and CROWE, C., (1988) An experimental and numerical study of flow and convective heat transfer in freely falling curtain of particles. *Journal of Fluid Engineering - Trans ASME*, **110**, 172-181.

LEE A (2009) Modelling the solids transport phenomena within flighted rotary dryers. PhD thesis, School of Engineering and Physical Sciences, James Cook University, Townsville.

LEE, A. and SHEEHAN, M.E., (2010), “Development of a Geometric Flight Unloading Model for Flighted Rotary Dryers”, *Powder Technology*, **198**, 395-403

MATCHETT, A.J. and BAKER, C.G.J. (1987) Particle residence times in cascading rotary dryers Part 1 - Derivation of the two-stream model. *Journal of Separation Process Technology*, **8**, 11-17.

ROJAS, R., PIÑA, J., BUCALA, V., (2010), "Solids transport modeling in a fluidised drum granulator", *Ind. Eng. Chem. Res.*, **49**, 6986-6997

SCHILLER, L. and NAUMANN, Z., (1935), “A Drag Coefficient Correlation”, *Zeitschrift des Vereines Deutscher Ingenieure*, **77**, 318-320

SHEEHAN, M.E., BRITTON, P. and SCHNEIDER, P.A., (2005), “A Model for Solids Transport in Flighted Rotary Dryers Based on Physical Considerations”, *Chem. Eng. Science*, **60**, 4171-4182

WARDJIMAN, C., LEE, A., SHEEHAN, M. and RHODES, M., (2008), “Behaviour of a Curtain of Particles Falling Through a Horizontal Gas Stream”, *Powder Technology*, **188(2)**, 110-118

WARDJIMAN, C., LEE, A., SHEEHAN, M. and RHODES, M., (2009), “Shape of a Particle Curtain Falling in Stagnant Air”, *Powder Technology*, **192(3)**, 384-388