

CFD study of droplet atomisation using a binary nozzle in fluidised bed coating

W. Duangkhamchan^{a,b}, F. Ronsse^a, F. Depypere^b, K. Dewettinck^b, J.G. Pieters^{a,*}

^a Biosystems Engineering, Faculty of Bioscience Engineering, Ghent University, Coupure Links 653, B-9000 Ghent, Belgium

^b Laboratory of Food Technology and Engineering, Faculty of Bioscience Engineering, Ghent University, Coupure Links 653, B-9000 Ghent, Belgium

ARTICLE INFO

Article history:

Received 11 August 2011

Received in revised form

6 October 2011

Accepted 9 October 2011

Available online 14 October 2011

Keywords:

Fluidisation

Computational fluid dynamics

Multiphase flow

Powder technology

Two-fluid atomisation

Population balance model

ABSTRACT

An Eulerian–Lagrangian computational fluid dynamics (CFD) model was built to describe two-fluid atomisation in a tapered fluidised bed coater using the air-blast/air-assisted atomiser model. Atomisation was modelled both with and without the inclusion of the solid phase (i.e. gas–liquid and gas–solid–liquid multiphase modelling). In addition, a multi-fluid flow model (Eulerian–Eulerian framework) combined with a population balance model was used as an alternative approach for modelling the spray produced by a two-fluid nozzle. In this approach, the CFD solver couples the population balance equation along with the Navier–Stokes equations for predicting the droplet diameter and mass fraction distribution. Comparison between simulated spray pattern (gas–liquid model) and that experimentally visualised by means of UV illumination was made and a good agreement was obtained. Parametric studies were done in order to investigate the effects of operating conditions on spray cone and liquid mass fraction inside the reactor. Furthermore, comparison of time-averaged fluidised bed behaviour with the inclusion of sprays obtained by both gas–solid–liquid multiphase modelling methods is presented.

© 2011 Elsevier Ltd. All rights reserved.

1. Introduction

Among a wide range of microencapsulation techniques, fluidised bed technology has been successfully used for the coating of particulate solids due to its excellent mixing capabilities and its optimal heat and mass transfer rates (Ronsse et al., 2007a,b). Fluidised bed coating is an added-value technique whereby a pure active ingredient or mixture of ingredients, in solid particulate form, is encapsulated within a coating polymer. The aim of encapsulation is to control release, to protect the core ingredients, to increase the overall product quality and to increase the processing convenience. An aqueous or organic solvent-based solution containing the coating polymer is continuously sprayed by means of a pneumatic or two-fluid nozzle, which may be submerged in or positioned above the bed (Depypere et al., 2009; Ronsse et al., 2007b). In top-spray configuration, regarded as the most appropriate method to be successfully used in the food industry due to its high versatility, relatively high batch size and relative simplicity (Depypere et al., 2009), the two-fluid nozzles are usually positioned above the bed, producing sprays of an aqueous solution of the coating material with a droplet size

ranging from 10 to 40 μm in order to coat particles (Hede et al., 2008; Ronsse et al., 2007b).

In top-spray fluidised bed coating, the basic operating principle consists of air suspension of particles in the coating chamber, spraying of coating polymer solution as droplets with the objective of increasing the probability of particle–droplet impact, spreading of droplets on the particle surface, droplet evaporation and layering or superposition of droplets on the particle surface resulting in a homogeneous coating enveloping the core particles (Teunou and Poncelet, 2002). In order to control process efficiency in fluidised bed coating using a model-based approach, it is necessary to explore each phenomenon taking place in the system. As described in previous works (Duangkhamchan et al., 2010; Duangkhamchan et al., 2011), the momentum transfer between the gas and solid phases was first modelled using various drag coefficient models, in order to evaluate the appropriate drag model for the description of fluidised bed behaviour (Duangkhamchan et al., 2010). However, in that work, only interaction between gas and solid phases with the absence of atomisation was taken into account. Subsequently, the solids volume fraction was simulated including the effect of the release of compressed air by the two-fluid nozzle in order to provide qualitative and quantitative consistency of model simulations with the experimental data (Duangkhamchan et al., 2011). However, the liquid phase, being the sprayed droplets, was not yet included in the latter study. Therefore, the next step – as outlined

* Corresponding author. Tel.: +32 9 264 61 88; fax: +32 9 264 62 35.
E-mail address: Jan.Pieters@UGent.be (J.G. Pieters).

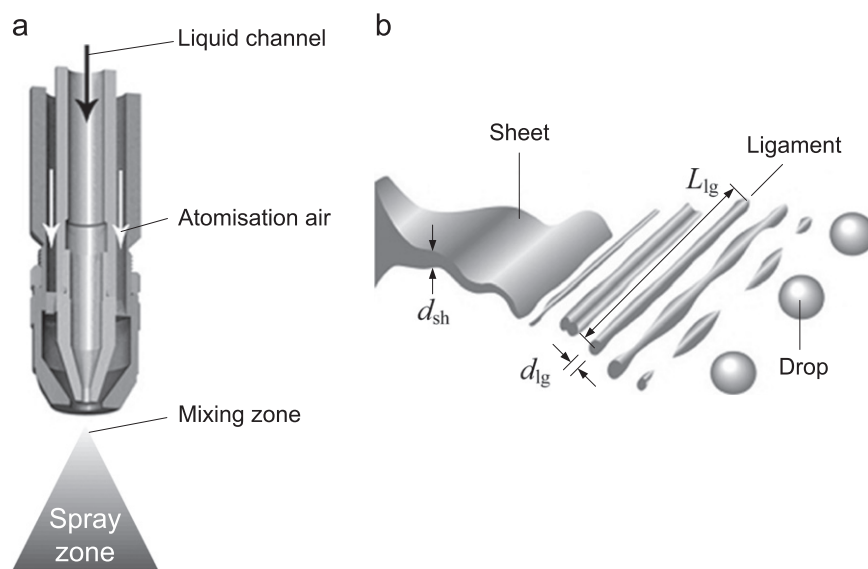


Fig. 1. Schematic overview of the two-fluid nozzle (a) and the mechanism of droplet atomisation (b), adapted from Salman et al. (2007) and Spray Drying Systems Co. (2000).

in this research article – is the addition of the liquid phase to the existing fluidised bed CFD model.

To produce sprays in fluidised bed coating processes, pneumatic or two-fluid atomisation is frequently used. In the mechanism of the two-fluid atomisation, as shown in Fig. 1, a high velocity gas impacts a liquid jet issuing from a nozzle orifice creating high shear force over the liquid surface, leading to disintegration into spray droplets. The optimum frictional conditions resulting from high relative velocity between gas and liquid are generated by expanding the air to sonic or supersonic velocities before impacting the liquid (Hede et al., 2008). When injected from the nozzle orifice, the liquid jet starts to make contact with the mixing zone, expanding radially and squeezed into a thin circular sheet (Zeoli and Gu, 2006). The term “liquid sheet” is used for both flat and cylindrical jets as common nomenclature (Hede et al., 2008). For more details about two-fluid atomisation, the reader is referred to Hede et al. (2008), Sridhara and Raghunandan (2010) and Varga et al. (2003).

Currently, to design and optimise the fluidised bed coating process, spray conditions and the operation of the two-fluid nozzle are identified as one of the most critical factors for the whole process and in practice have to be trial-and-error tested in order to control spray characteristics, including droplet size distribution, droplets trajectories and spray cone angle (Hede et al., 2008; Ronsse et al., 2007b). Therefore, in order to reduce time consumption and expensive cost of extensive experiments, many numerical approaches, for instance, Eulerian–Eulerian CFD, Eulerian–Lagrangian CFD, population balance modelling, etc., have been developed as a powerful tool to comprehend or clarify the impact of different input variables on process efficiency and to research and design work (Ronsse et al., 2007b).

During the last few decades, CFD has been widely adopted in many industrial uses. In spray application, various numerical methods, for instance, the volume of fluid (VOF) method and the discrete phase method (DPM), have been developed to predict basic characteristics of spraying nozzles (e.g., spray angle and droplet size distribution) and to predict droplet trajectories. In the discrete phase method (Lagrangian framework), the droplet trajectory is calculated individually using the equation of motion, whereas the volume of fluid or multi-fluid method (Eulerian framework) is based on continuum mechanics which treat the two phases as interpenetrating continua (Taghipour et al., 2005).

For instance, for studying the two-fluid atomisation, instead of using only experimental PIV to provide an instantaneous map of the entire velocity field, Hoeg et al. (2008) used Eulerian CFD models to investigate the flow pattern of gas and liquid jets issuing from a two-fluid nozzle. In that work, good agreement between model-predicted and experimental data was found. Furthermore, Zeoli and Gu (2006) used the discrete phase model to simulate the critical droplet breakup during atomisation producing fine spherical metal powders. To verify their model performance, the liquid metal was initialised to large droplet diameters varying from 1 to 5 mm. They found that the model could provide quantitative assessment for the atomisation process. Pimentel et al. (2006) improved the capability of CFD models to capture liquid atomisation mechanisms of the two-fluid nozzle associated with the measured droplet diameters to initialise the droplet size in the discrete phase model. Even though many researchers have attempted to model droplet atomisation in various applications by means of CFD, as seen in Behjat et al. (2010), Fuster et al. (2009), Gianfrancesco et al. (2010), Kalata et al. (2009), Mezhericher et al. (2010), White et al. (2004) and Yamada et al. (2008), two-fluid atomisation occurring in the fluidised bed coating process still needs to be explored, considering the fact that the liquid is atomised in the presence of the fluidised solid phase.

In addition to the two approaches for modelling multiphase flow problems mentioned previously, the population balance model has been introduced to the CFD community as an alternative approach because of its reduced level of computational complexity (Aly et al., 2009). Moreover, the model can be easily coupled with the Eulerian–Eulerian model which eliminates the need for semi-empirical models employed in the Lagrangian framework (Aly et al., 2009). Recently, the population balance model has been extensively used in liquid–liquid and gas–liquid systems for modelling droplets and bubbles (Aly et al., 2010a,b). However, only few studies can be found in droplet atomisation problems, especially in fluidised bed coating systems. The atomisation process occurring in a plain jet air blast atomiser (two-fluid nozzle) was first investigated using the combination between a population balance model and a CFD Eulerian multi-fluid model by Aly et al. (2009). In that work, although the model obtained good agreement with experimental data, improvement still needed to be done. Therefore, Aly et al. (2010a,b) developed a new mathematical model for calculating droplet breakup

Download English Version:

<https://daneshyari.com/en/article/10262482>

Download Persian Version:

<https://daneshyari.com/article/10262482>

[Daneshyari.com](https://daneshyari.com)