TRANSIENT SIMULATION OF AIR FLOW IN A CO-CURRENT SPRAY DRYER

SAAD NAHI SALEH

Chem. Eng. Department, Tikrit University
Baghdad, Iraq
Tel.:009647710640208, E-mail:saad_nahi68@yahoo.com

Abstract: This paper simulates numerically the air flow behaviour with no swirl entry in a co-current pilot plant spray dryer, using a transient three-dimensional Reynolds-average Navier-Stokes equations, closed via the RNG $k-\varepsilon$ turbulence model, solved using the Fluent 6.3 commercial CFD code. Self-sustained oscillating central jet of air is predicted. The simulated air flow jet reveals flapping rather than precession motion. The main frequency of flapping of the central jet is quantitatively assessed. The results are shown to be in good agreement with experimental data from the literature.

Keywords: Spray dryer, CFD, Air flow, Simulation.

INTRODUCTION
Spray dryer equipment have a wide range of important industrial applications such as chemical, food, and pharmaceutical industries to produce a powder product with specific properties such as moisture content, bulk density, particle size distribution, etc… In addition to above specification, performance of the spray dryer is acceptable with achieving another group of parameters, including minimal wall deposition, uniform residence time distribution, maximum energy efficiency and minimum thermal decomposition. These objectives are significantly affected by the prevailing air flow pattern (Southwell & Langrish, 2000).

Experimental and numerical investigations of air flow in co-current pilot plant spray dryer at no swirl proved that air flow consist of the central air core bounded by slow recirculation zones (Oakley et al. (1988), Le Barbier et al. (2001), etc…). Bigger particle sizes penetrated the core and stalled in the recirculation zones for more time which lead to hitting the wall or depositing if it in the sticky case (Kieviet, 1997).

Numerous experimental and numerical studies have been published for analysis of the air flow pattern in the spray dryer. Numerical analysis such as Computational Fluid Dynamics (CFD) of air flow pattern has proven to be a very useful tool to predict the flow behaviour in complex geometries of spray dryers (Fletcher et al., 2006). CFD have been used to analyze the performance of an industrial spray dryer in advance of making major structural changes to the dryer, and have submitted important proposed changes which greatly improve the quality of the product (Fry, 2001).

For the co-current spray dryer, there are generally two configurations to discharge the product from the dryer chamber (Masters, 1985): primary product discharge (Fig.1 Type 1) and totally discharge product (Fig.1 Type 2). In first configuration, one of the arrangements of the outlet is a pipe mounted through the wall of the cone section of the chamber, bent downwards in the centre of the chamber. The downward facing exhaust pipe has a minimum interference on the air pattern, and in fact, acts to maintain the pattern down into the chamber cone.

Fig. 1 Conventional product discharge types from conical chamber bases
Type 1 Primary product discharge
Type 2 Total product discharge

The air flow in first configuration was simulated by using three dimensions CFD model which demonstrate necessity to consider the transient behaviour of the air flow in some locations in the chamber which the experimental measurements reveal it (Saad Nahi, 2010).
The configuration Type 2 has been studied more extensively than Type 1. Most of the experimental and numerical investigations found that air flow pattern revealed some oscillations in this configuration (Type 2) (Langrish et al., 2004).

The main objective of this paper is to examine numerically the air flow with no swirl entry in first configuration (Type 1) and evaluate the effects of the downward bent outlet on the air pattern using transient three dimensions CFD model and comparing it with the published experimental data.

Fig. 2 The geometry of the pilot plant spray dryer (the dimensions in mm)

NUMERICAL MODELLING
The computational fluid dynamic CFD package FLUENT 6.3 has been employed to implement the three-dimensional transient model of air flow in the spray drying chamber. The pilot-scale co-current spray dryer used in this simulation has the dimensions shown in Fig. 2 (Kieviet, 1997). Three-dimensional model geometry is chosen since the two-dimensional, axisymmetric simulations do not reproduce the basic physics that are involved (Langrish & Fletecher, 2003). A fully three-dimensional unstructured mesh as shown in Fig. 3 has been constructed using 516000 tetrahedral mesh elements. The Reynolds-averaged Navier-Stokes equations have been solved, with the standard RNG $k-e$ turbulence model being used (Fluent, 2006). The inlet boundary conditions of the simulated case correspond to the experimental conditions with no inlet swirl which is conducted by Kieviet (1997) with the same dryer. These quantities were measured and were found to be the velocity vector (axial, radial, tangential) = (6.03, -4.22, 0.00 m/s); the turbulent kinetic energy ($k$) = 0.027 m$^2$/s$^2$, rate of dissipation ($\varepsilon$) = 0.37 m$^2$/s$^3$.

The numerical model was initially solved using the steady state calculation until convergence. The solution of the steady state was then used as the initial condition for the unsteady case. A time step of 0.02 s was adopted. Typically 40-70 iterations led to a convergent solution at each time step with local residuals of less than 10$^{-6}$.

Fig. 3 The mesh used in the simulations

RESULTS AND DISCUSSION
Transient simulations of air flow with no inlet swirl in co-current spray dryer are analyzed. Simulated instantaneous velocity contours (Fig.4) from transient simulation at 185, 188, 191, 194 and 197 s for the selected case show that the major flow feature is a main axial jet which broadens as it moves further into the chamber. Because of the internal exit bent pipe which reduces the area for air to go through at one side of the drying chamber (Haung et al., 2006), the central jet is offset from the geometric centre and deflects towards this side of the conical wall. Then a part of the jet separates and moves towards the opposite side of the jet deflection. It reattaches the conical wall and flows upstream along the wall, then traverses the chamber in the radial direction creating a particular large eddy pattern of air recirculation related to the quantity of separated proportion of the jet and the reattachment point on the wall. This behaviour is schematically interpreted in Fig. 5.

The velocity counters as shown in Fig. 4 indicate the time-dependent nature of the flow patterns. Therefore, it is very sufficient to consider the overall flow structure of present case as unsteady flow.
By referring to snap shots of simulated instantaneous velocity contour plots at a height of 0.6, 1.3, 2.0, 2.7 m from the ceiling of the spray dryer (Fig. 6), it is showed that there is a an irregular cyclic behaviour of air flow where the recirculation and traversal flow create an unbalance pressure in the regimes surrounding the jet and then developing a cross flow which provides a coupling between the fluid in these regimes and consequently deflects of the jet. This cycle continues to produce a self sustained oscillation of the jet. This mechanism which interprets the transient behaviour of air flow coincides with the numerical investigations of Gebert et al. (1998) for a confined submerged liquid jet. This description is also consistent with the experimental work of Kieviet (1997) who observed this flow feature as a fast flow core wiggled in space. In these simulation times considered, the air jet flow exhibits cyclic behaviour generating a self-sustained oscillation. The period of oscillation is 13 sec.

For studying the periodicity of the flow structure, Fast Fourier Transform (FFT) analysis was undertaken to translate velocity-time signal data sampled in certain time duration into the frequency spectrum for a monitoring point at a height of 1.14 m from the ceiling of the dryer and 0.57 m from the geometric centre. The frequency spectrum (Fig. 7) showed a dominant frequency of 0.075Hz, which corresponds to a period of 13s. This period is similar to that found by experimental investigation of Kieviet (1997) in this spray dryer who noted that a slow periodic fluctuation of the velocity was recorded at this location and the characteristic time of this fluctuation was approximately 12 s.

It is widely accepted in the literature that an oscillation mode is characterized by a Strouhal number where the frequency data were scaled with the expansion ratio $E$ (chamber diameter to inlet diameter) and the inlet jet velocity to form the Strouhal number as defined by Hill et al. (1998):
Fig. 6 Velocity contour plots at a height of 0.6, 1.3, 2.0, 2.7 m from the ceiling of the spray dryer.

\[ S_f = \frac{2}{\sqrt{\pi}} E^2 \int_0^U \]  

(1)

For the current expansion ratio of 4.46 and aspect ratio \( L/D \) (total length to chamber diameter) is 1.7, the FFT spectra (Fig. 7) shows a two large peaks of frequency (0.075, 0.04), corresponding physically to the flapping and the precession oscillation of the central jet. The finding of the two characteristic frequencies in term of the Strouhal number (0.114, 0.06) is consistent with the observation of Lebarbier et al. (2001) for a spray dryer of the similar geometrical proportions (Type1). Guo et al. (1999) noted that the both the flapping and the precession occur within the range of expansion ratio \( E=3.95-6.0 \) for sudden expansion pipe configuration. A further increase in the expansion ratio produced a processing oscillation as simulated by Langrish et al. (2004) for a spray dryer (Type 1) with \( E=7.27 \). The current flapping frequency in term of the Strouhal number (0.114) is also consistent with that for the case of a sudden expansion with downstream contraction of \( E=3.77 \) and \( L/D=3 \) (Gue et al., 2003). This interprets why Kieviet (1997) used the wiggling (not precession) term to describe the movement of the central jet through his investigation of the air flow pattern in a co-current spray dryer (Type1). Therefore the flapping motion of the air flow jet predominates in the geometric configuration investigated here.

It is noteworthy that previous studies about the air flow pattern without inlet swirl have proved the stability of the air jet was affected by inlet velocity and geometry configuration such as expansion ratio and aspect ratio. Woo et al. (2009) predicted that the dominant frequency was 0 Hz, were the air flow pattern was no long scale flow structure in the spray dryer of totally discharge product configuration (Type 2). The simulated jet flow pattern for present case (Type 1) as shown in Fig. 4 and Fig.6 indicates that the jet weakens and oscillates in the conical part of the dryer. This is consistent with video analysis conducted by Lebarbier et al. (2001) in dryer (Type 1) which exhibited a central jet that precessed around the axis with time where precession motion of the central jet for no inlet swirl was limited to only the bottom of the spray dryer. These findings demonstrate that chamber (Type 1) exhibits a central jet that oscillates with time due to the maldistribution of the jet in the conical part of the chamber. Guo et al. (2003) noted that the downstream contraction is expected to stabilize the flow instability, and thus probability of precession is less than that of flapping. Therefore any slight modification in the dryer configuration especially in the conical part of the short dryer whether the air outlet or contraction this part may result in a transient air flow.
CONCLUSIONS
It is concluded that the air flow pattern was largely time dependent. The predicted air flow by transient three dimensions CFD model provided a quantitative depiction about the global flow pattern of air in the spray dryer.
In addition to the effects of the geometry configuration of the dryer and the inlet velocity of the air on the air flow behaviour inside the dryer, the air outlet channel configuration can be considered a significant factor which plays an important role in behaviour of the air flow. When the configuration is slightly asymmetrical, the air jet can produce an oscillatory motion. The flapping motion predominates in the geometric configuration investigated here.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>d</td>
<td>inlet diameter</td>
<td>m</td>
</tr>
<tr>
<td>E</td>
<td>expansion ratio</td>
<td>-</td>
</tr>
<tr>
<td>f</td>
<td>frequency</td>
<td>Hz</td>
</tr>
<tr>
<td>St</td>
<td>Strouhal number</td>
<td>-</td>
</tr>
<tr>
<td>U</td>
<td>inlet velocity</td>
<td>m/s</td>
</tr>
</tbody>
</table>

REFERENCES