NCNE-2020 Special Issue

J. Indian Chem. Soc.,
Vol. 97, July 2020, pp. 1142-1145

Computational investigation of air solid flow in a spray dryer for effluent treatment

Sunil Kumar Singh and B. Ashraf Ali*

[a]Department of Chemical Engineering, National Institute of Technology Karnataka, Surathkal-575 025, Karnataka, India

E-mail: ashrafmchem@gmail.com

Manuscript received online 20 April 2020, accepted 15 June 2020

In this work, the hydrodynamics and evaporation rate of the co-current spray dryer is numerically investigated through ANSYS Fluent (CFD). The performance of the spray dryer depends on the geometry, operating conditions, and underlying hydrodynamics in such systems. To predict the air-solid flow in a spray dryer, the Euler-Lagrangian CFD model is used to track the particles in the dryer. The continuous phase turbulence is predicted using RNG version of k-turbulence model. To quantify the flow pattern, a horizontal line is considered and spatial variation of velocity profiles are analyzed. The predicted air velocity variation was found to be maximum at the center of the core. Further, the airflow pattern is analyzed for various operating temperatures and feed properties. It was found that airflow pattern influences particle behavior with minimum deposition rates on each section of the wall when air temperature is 350 K.

Keywords: Euler-Lagrangian, CFD, hydrodynamics, evaporation, spray dryer.

Introduction

In pharmaceutical industries, the reject wastewater from the reverse osmosis plant contains concentrated dissolved harmful chemicals. These chemicals cannot be directly discharged into the freshwater bodies and have to be treated. The untreated reject adversely affects the aquatic life and the chemicals are to be separated out. Conventionally, the wastewater is concentrated in multiple effect evaporators and crystallized out. This process is cost-intensive and increases the cost of the effluent treatment plant. In this regard, a cost-effective process equipment is needed that requires minimum amount of energy in converting dissolved solids to dry powder. Spray drying is a process that involves the removal of volatile component for producing dry powder to get the dispersed phase separated out from the liquid. This is indeed a complex process due to combination transport of particles within the dryer and the simultaneous removal of moisture process. In contrast to traditional approach, CFD simulations helps to visualize and to predict simultaneous phenomenon experimented by the particle in spray dryer. The performance of the spray dryer depends on the flow patterns and temperature distributions of air and particle.

Cher *et al.*[1] performed CFD simulation to investigate the hydrodynamics of spray dryer for three different turbulence models (Standard, RNG and realizable k-). They found that standard k-predicts accurately continuous phase turbulence when swirling is not significant and RNG k- for swirl flow. Saad Nahi Saleh[2] found that the air circulation rate, feed velocity, air velocity and size distribution of solid particles influences final dried product quality. Rajashekhara *et al.*[3] and Muzammil Ali *et al.*[4] found that air flow pattern, moisture content of particle, size and its velocity influence the deposition rate of solid particle on the dryer surface. In most of the published CFD literature on spray drier are focused on predicting the air temperature and its velocity profiles using Euler-Lagrangian approach. The histories of dispersed phase particle such as residence time, velocity, temperature are completely ignored in most of the literature. Moreover, understanding of trajectories of particles and its impact on walls of spray dryer is important because it influences bulk density of dried powder (ex. heat sensitive materials like paracetamol). This significantly affects flowability of solids and eliminates fine particles, hence performance of spray dryer decreases.

Hence, in the present work, these effects are considered to find optimum condition that supports air circulation rate and drying characteristics.

The article is organized by describing geometry of spray dryer at first. Then computational models and simulation methodology are discussed. The numerical predictions are analyzed in Results and discussion section. Finally, the important findings are reported.

Geometry of spray dryer:

The geometry of the co-current spray dryer is depicted in Fig. 1. It consist of a cylindrical section of 224 mm diameter and 205 mm height. The height of cone is 175 mm. Air is introduced in the annulus and it is co-current to the feed inlet. The hydraulic diameter of air inlet is 43 mm. The effluent slurry (paracetamol) is injected through a nozzle of diameter 0.2 mm and at an angle of 45° from the top of the column.

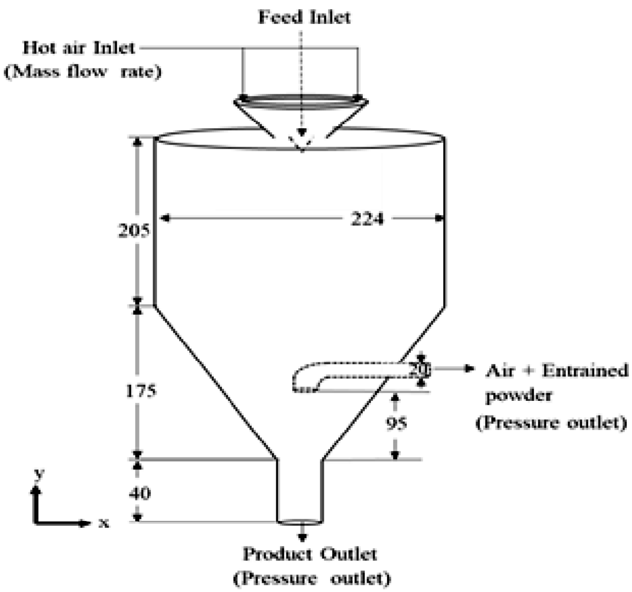

Fig. 1. Schematic of the spray dryer (mm).

Computational modeling:

The hydrodynamics in a spray dryer fitted with nozzle is numerically investigated using Euler-Lagrangian model (Alessandro *et al.*[5]). Here hot air is modelled as continuous phases and feed (paracetamol) is treated in Lagrangian manner. The trajectories of the dispersed phase are computed by solving Newton's second law of motion in Lagrangian reference frame. The species transport model used along DPM model is used to predict simultaneous transfer of heat and mass characteristics during the process of spray drying. The transport equations and other details are obtained from literature (ANSYS Fluent, Theory Guide, 2019)

Simulation methodology:

To predict flow and characterize drying of paracetamol, 3D transient CFD simulations are performed using ANSYS 2019 R3. The geometry is created using ANSYS workbench design modular. Grid independence study were carried out with increasing number of cells such as 1,00,000, 3,00,000, and 5,00,000. It is found that the results are invariant above 3,00,000 and hence it is considered to be optimum. At the hot air inlet is modeled as mass flow rate boundary condition (BC) and at the outlet pressure outlet BC is specified. No slipboundary is specified at all the stationary walls of the spray dryer. The CFD simulations are performed using discrete phase model with optimized time step 0.01s.

Results and discussion

To investigate the flow patterns in a spray dryer, iso-contours of air velocity is analyzed at Z = 0 mm. This is shown in Fig. 2. It is observed that magnitude of velocity of air increases as time progresses with maximum velocity at the center with a low velocity region around it.

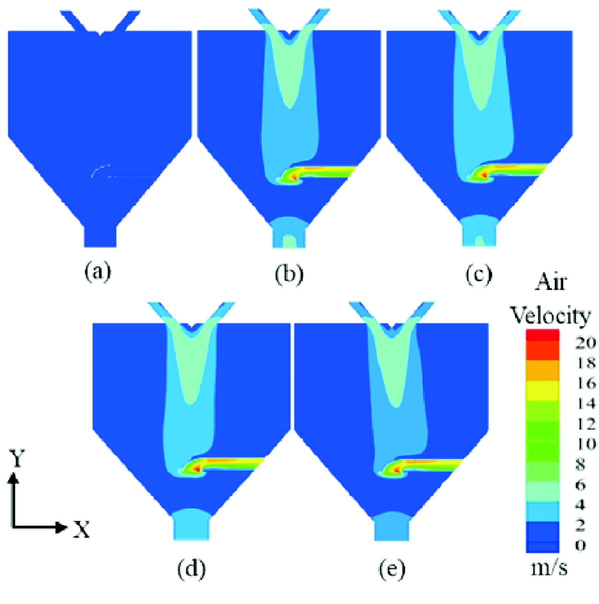

Fig. 2. Iso-contours of air velocity at (a) 0s, (b) 0.5s, (c) 1s, (d) 3s and (e) 5s.

A horizontal lines are chosen along the height of spray dryer to quantify the spatial variation of air velocity. This is depicted in Fig. 3. These lines are chosen along different heights of the spray dryer. At the core region, air velocity is observed to be maximum and its magnitude decreases move downwards in the spray chamber. The predicted air flow pattern is found to be symmetric at 15 mm (top) and asymmetric at the bottom section of spray dryer. This is attributed to the area of flow decreases from top to bottom.

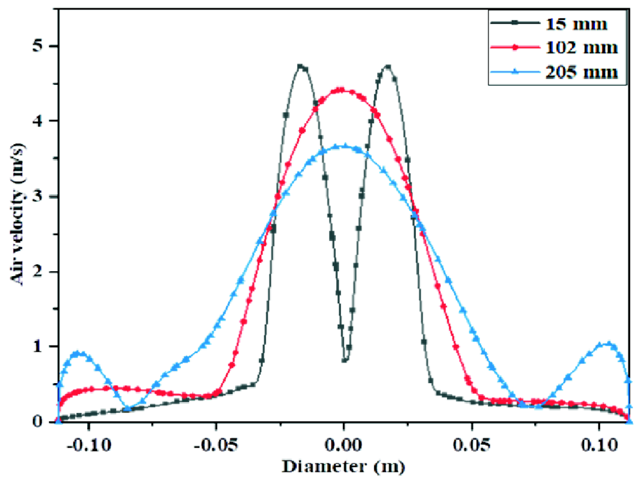

Fig. 3. Spatial variation of air velocity.

The predicted flow field is quantified further by calculating the air circulation rate[6]. This is calculated by

$$\int \rho_{da} u_{da} dA$$

where ρ is the density of dry air, *u* is the velocity of dry air and *dA* is the differential surface area, in the flow domain. To calculate this iso-surface is chosen Y = 0m near the cylinder-cone junction. The air circulation rate is calculated for various air temperaturesby varying densities of feed mixture. This is shown in Fig. 4. It is observed that the increase in the densities of the feed mixture offers significant resistance in the air flow and hence air circulation rate decreases. The air circulation rate is found to be maximum at 350 K for all densities of feed mixture, thus 350 K is considered to be an optimum. To validate this temperature further, the predicted particle residence time (PRT) are analyzed by varying air temperature. This is shown in Table 1. The PRT is computed by tracking the number of particles (feed) through the flow

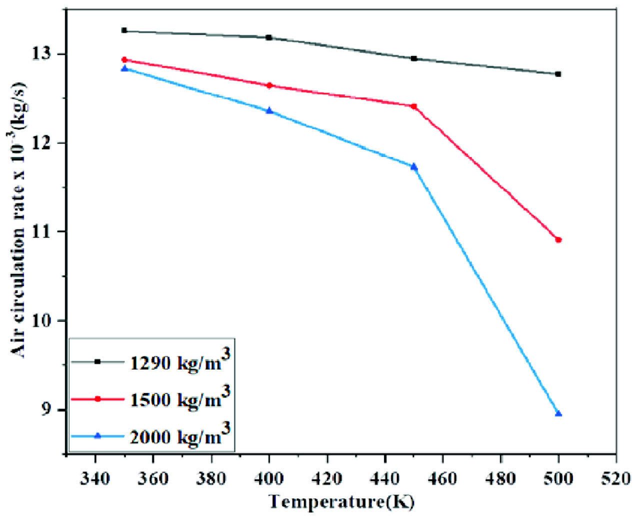

Fig. 4. Effect of air temperature and feed density on air circulation rate.

Table 1. Effect of air temperature on particle residence time and the relative velocity between feed and air

Air temperature (K)	Particle residence time $\times 10^{-2}$ (s)	Relative velocity (m/s)
350	3.79	0.51
400	3.76	0.67
450	3.55	0.91
500	3.44	1.02

domain and it represents, the time spent by the particles in the drying chamber. This in turn depends on air flow pattern. It is observed that the PRT decreases with increase in air temperature and found to be relatively high when air temperature is 350 K. This is attributed to lower value of relative velocity magnitude between the phases. Thus the observed predictions once again confirms the optimized temperature (350 K).

To find an optimum air velocity at which spray dryer needs to be operated, CFD simulations are performed for various air velocities and its effect on air circulation rate is calculated. This is shown in Table 2. It is observed that as velocity of air influences circulation rate and its value is high for the air velocity of 6 m/s. This velocity is found to be optimum and considered for further investigation.

So far, spray dryer was analyzed without incorporating the evaporation model. Now, to predict evaporation rate in a

Table 2. Effect of air velocity on air circulation rate

Air velocity (m/s)	Air circulation rate $\times 10^{-3}$ (kg/s)
2	3.912
3	6.575
4	9.929
5	10.251
6	13.478

spray dryer, species transport equations are solved along with the continuity, momentum, and energy balance equation. Here CFD simulations are performed for various initial moisture contents of feed (1290 kg/m^3; 100 μm) at optimized air temperature and velocity of 350 K and 6 m/s respectively. The evaporation rate and particle deposition probability (%) at the various sections of the dryer surface are calculated and reported in Table 3. It is observed that the evaporation rate and amount of particle deposited on dryer surface increases as increase in the amount of initial feed moisture content as it has higher volatile substances. The amount of

Table 3. Effect of moisture content on evaporation rate and amount of particle deposition at the dryer surface

Moisture content (kg_w/kg_f)	Evaporation rate x10^{-4} (kg/s)	Particle stickiness on the wall (%)		
		Cylindrical	Conical	Ceiling
0.11	0.27	2.58	7.05	0.002
0.25	0.55	8.22	11.4	0.007
0.66	1.06	9.97	28.4	0.010
1.50	1.59	14.72	42.6	0.021
4.00	2.07	19.44	56.9	0.030
9	2.36	21.61	64.2	0.041

particle deposited on the conical surface is found to be larger in compare with cylindrical and ceiling surface. This is attributed to weak recirculation of air near the conical surface and strong recirculation of air near to the cylindrical surface. This in turn affects the quality of effluent. Less than one percentage of particles are noticed at the ceiling of the spray chamber, as expected. Hence it is advantageous to use very low air temperature (350 K) and lower amount of initial feed moisture, so that the air circulation rate is maximum, particle deposition on dryer surface is minimum to obtain the better product quality.

Summary and conclusions

The flow field in a spray dryer is investigated using Euler-Lagrangian CFD model. The spatial variation of predicted air velocity was found to be maximum at the core of spray dryer. CFD simulations are performed for various air temperatures and its circulation rate was calculated to optimized temperature. The calculated air circulation rate was found to be highest for 350 K. This is further verified by analysing the particle histories such as particle residence time (PRT) and relative velocity magnitude. Further, the predicted evaporation rate analysed for various amount of initial moisture content. The evaporation rate was found to be maximum when moisture content in the substance was high. Hence, this study concludes, 350 K likely to be suitable temperature for heat sensitive materials like paracetamol.

References

1. P. S. Cher, W. Rashmi, M. Khalid, C. H. Chong, M. W. Woo and L. H. Tee, *International Food Research Journal*, 2014, **21(3)**, 1089.
2. Saad Nahi Saleh, *International Journal of Chemical and Molecular Engineering*, 2010, **4(2)**, 226.
3. M. C. Rajashekhara and Raghavendra, *International Advanced Research Journal in Science, Engineering and Technology*, 2015, **2(6)**, 105.
4. Muzammil Ali, Tariq Mahmud, Peter John Heggs, Mojtaba Ghadiri and Andrew Bayly, *Drying Technology*, 2017, **35 (3)**, 281.
5. G. Alessandro, Christelle Turchiuli, Denis Flick and Elisabeth Dumoulin, *Food Bioprocess Technology*, 2010, **3**, 946.
6. R. W. Fox, J. T. McDonald and P. J. Pritchard, "Introduction to Fluid Mechanics", Wiley, 2003; B. Ashraf Ali and S. Pushpa Vanam, *International Journal of Multiphase Flow*, 2011, **37**, 1191.
7. L. Huang and A. S. Majumdar, *Drying Technology*, 2006, **24(6)**, 727.
8. Saad Nahi Saleh, A. H. Laith, *Iraqi Journal of Chemical and Petroleum Engineering*, 2016, **17(2)**, 69.