NCNE-2020 Special Issue

J. Indian Chem. Soc.,
Vol. 97, July 2020, pp. 1129-1132

Computational investigation of hydrodynamics and drying of industrial sludge waste in a spouted bed column

Santhosh Kumar N. and B. Ashraf Ali*

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

E-mail: ashrafmchem@gmail.com

Manuscript received online 21 March 2020, accepted 12 June 2020

The disposal of sludge from waste water treatment plants adversely affects the environment. Since sludge wastes are sticky, drying of such waste water sludge is challenging. Conventionally, these sludges are dried in open spaces where a large area of land is required which is a time-consuming process. To overcome this, spouted bed is used in the present investigation. The spouted bed is a gas-solid contactor for handling coarse particles of size greater than 1 mm with low operating pressure. In this work, hydrodynamics of waste water sludge in a conventional spouted bed is numerically investigated using Computational Fluid Dynamics (CFD). Euler-Eulerian CFD model is used to study the flow pattern in such system. The continuous phase turbulence (air) is modeled using standard κ - ε model. The spouting height and solid circulation rate are calculated to analyze spouting behavior. This is compared with a draft tube spouted bed system and found that the draft tube supports in enhancing the spouting characteristics of the bed. Further, an optimum draft tube configuration is found that promotes solid circulation rate. The drying characteristics are analyzed for various operating conditions and found that the temperature of air significantly improves the rate of drying.

Keywords: Spouted bed, sludge drying, CFD, draft tube, solid circulation rate.

Introduction

Sludge wastes generated from various industries causes problems when exposed in the environment because of the normal decomposition process. The removal of water content reduces the handling and transportation cost before disposal. A spouted bed is a gas solid contactor and it is used for drying such sludge wastes. In contrast to the traditional fluidization, here the gas is introduced through a nozzle at the base for spouting of the particles. This creates three well defined zones such as spout (carries solid particles upwards; gas dominated phase), fountain (carried solid particles forms a fountain and fall back into the bed) and annulus (slowly solid particles move downward; solid dominated phase). The entrainment of the solid particles in gaseous spout moves upwards. This forms a fountain of solid particles above the bed level and disengages into the bed surface. This leads to solid circulation in the bed and efficient mixing of coarse granular materials. Thus, the volume fraction of solid particles in the spouted bed column changes from spout to the annulus region. The spouted bed finds its applications in drying (Souza and Oliveira[1]), granulation (Shuyan *et al.*[2]), gasification (Lopez *et al.*[3]) and combustion (Konduri *et al.*[4]).

Several CFD studies on spouted bed column arereported in the literature[5–8] to visualize and predict detailed gas-solid flow field for scale up and optimization of spouted bed column.

To predict the flow pattern of gas-solid in the spouted bed column, the two fluid CFD models such as Euler-Eulerian (EE) or Euler-Lagrangian (EL) approaches are used. Duarte *et al.*[5] analyzed flow pattern in such system through Euler-Eulerian approach. In a draft tube spouted bed, the gas-solid pattern is predicted by Euler-Eulerian approach (Szafran and Kmiec[6]). Zhao *et al.*[7] analyzed the particle (granular) dynamics using a 2D axisymmetric spouted bed column with draft tube (DT) by discrete element modeling. Their simulation predictions are validated with PIV measurements. Their results indicate that particle velocity does not significantly affect spouting behavior and entrainment height, although it has a great effect on solid circulation rate. San Jose *et al.*[8] predicted the average cycle time and found that its magnitude decreases with increase in gas inlet diameter for the treatment of sludge from the paper industry.

From the literature review, it was found that the effect of solid circulation rate on the performance of spouted bed column is not studied. Hence, in the present work, these effects are considered using 3D Euler-Eulerian CFD model. The effect of spouting behavior and geometry of draft tube on solid circulation rate are analyzed in this article. Also, the significance of draft tube configuration on spouting and drying characteristics are addressed. The manuscript describes the spouted bed column at first. Then, the computational model and simulation methodology are discussed. Subsequently, the CFD results are discussed and concluded with the findings.

Geometry of spouted bed:

A cylindrical spouted bed of diameter 15.2 cm with conical bottom is considered for numerical investigation. The total height of the column is 72.5 cm. To obtain solid circulation rate,cone angle of 60° is maintained (Souza and Battega[9]). The diameter of the gas inlet is 1.9 cm. This is shown in Fig. 1.

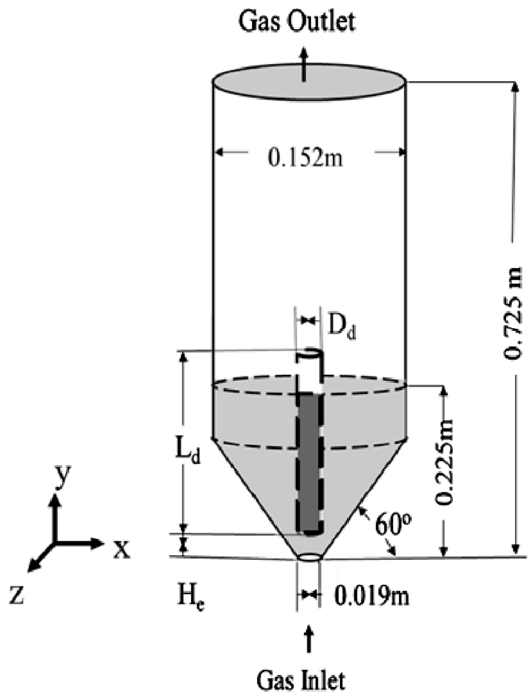

Fig. 1. Geometry of draft tube spouted bed.

Computational modelling:

To investigate flow field and to characterize drying in spouted bed, Euler-Eulerian approach is used. A CFD software package (Ansys Fluent v19.2) is used to predict the flow and characterize drying. To model interphase mass transfer, two phase drying rate is considered. The falling drying rate is predicted using the solution of particle diffusion with constant surface moisture. The transport equations and other details are obtained from the literature (Ansys Theory Guide 19).

Simulation methodology:

To analyze the flow pattern, transient simulations are carried out using the commercial software package (Ansys Fluent v19.1). The spouted bed geometry is developed through Ansys workbench. The CFD predictions checked using grid independence studies and 3,00,000 cells found to be optimum for detailed investigation. The gas inlet is modeled using velocity inlet boundary condition (BC) and pressure outlet is specified at the outlet (Ashraf Ali *et al.*[10]). All the stationary walls of the column are modeled as no-slip BC. The pressure and velocity fields are coupled using SIMPLE algorithm. A transient simulation isperformed using Δt = 0.001s. The convergence criterion (10^{-6}) is specified for each scalar and vector.

Results and discussion

The flow behavior in a conventional spouted bed (CSB) has been analyzed at first. To predict gas-solid flow, 3D CFD simulations are performed. The initial static bed height is 22.5 cm (particle diameter = 0.15 cm and density = 2500 kg/m^3) maintained for all CFD analysis with finite amount void fraction of solid 0.63. The vertical plane at Z = 0 is chosen to analyze the flow pattern. The contours of volume fraction of solids are analyzed for different lengths of the draft tube. This is depicted in Fig. 2.

Steady (15 cm and 20 cm) and unsteady (10 cm and 25 cm) spouting behavior are observed. The periodic flow fluctuation in the gas solid flow affects the draft tube length such as 10 cm and 25 cm and hence unsteady spouting are found.

CFD simulations are performed for different gas velocities and its effect on spouting height is calculated to find optimum draft tube length that supports spouting behavior. This is shown in Fig. 3. The distance between spout orifice and fountain top surface is treated as spouting height.

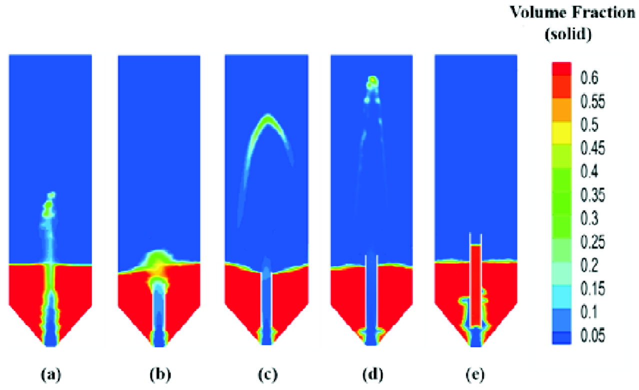

Fig. 2. Iso-contours of solid volume fraction: (a) CSB, (b) L = 10 cm, (c) L = 15 cm, (d) L = 20 cm and (e) L = 25 cm.

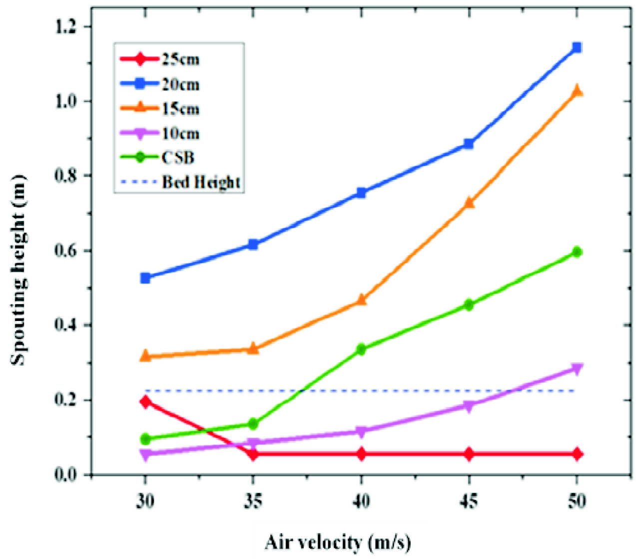

Fig. 3. Effect of draft tube length on height of the spout.

The spouting height is observed to be increasing as the gas velocity increases for all draft tube lengths. The calculated spouting height is maximum for 20 cm draft tube length. Hence, this draft tube length (20 cm) is considered for further investigation.

To further quantify the flow field, solid circulation rate (Thorley *et al.*[11]) is calculated using

$$W \cong V_s A_A \rho_s (1 - \epsilon_A) \quad (1)$$

Here, ϵ_A denotes the void fraction of solid within annulus and A_A is the cross-sectional area of annular region. This is calculated for different gas velocities and shown in Fig. 4.

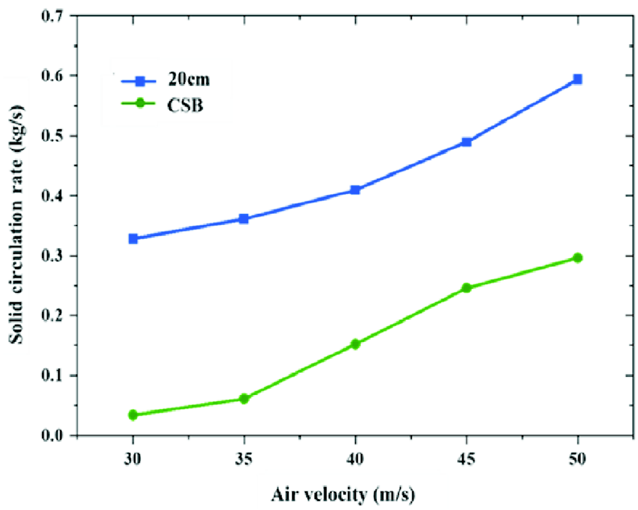

Fig. 4. Effect of draft tube (L = 20 cm) on circulation rate of solid.

The solid circulation increases with increase in gas velocities and once again it was found that 20 cm draft tube length is more advantageous in characterizing spouting behavior. Hence, this draft tube length is considered to be optimum.

Further effect of entrainment height on solid circulation is calculated for the optimized draft tube length (20 cm) and it is reported in Table 1. Here, the height difference between the spout orifice and the draft tube inlet is considered to be entrainment height (H_e).

Table 1. Effect of entrainment height H_e on circulation rate of solids

Entrainment height (H_e) (cm)	Solid circulation rate (kg/s)
4	0.82
5	0.49
6	0.28

The calculated solid circulation rate is found to be higher when entrainment height is 4 cm and spouting behavior is observed only when $\frac{H_e}{D_s} = 10$ (where D_s is diameter of solid).

CFD simulations are performed for different draft tube diameter (for optimized draft tube length, 20 cm and H_e, 4 cm) and its effect on solid circulation is calculated. This is reported in Table 2.

Table 2. Solid circulation rate for different draft tube diameter

Draft tube diameter (cm)	Solid circulation rate (kg/s)
2.0	0.41
2.2	0.85
2.4	0.66

Here, the solid circulation rate is observed to be highest for draft tube with 2.2 cm diameter. Thus, the hydrodynamics helps to find the optimized draft tube configuration that supports spouting behavior in spouted bed column.

This optimized draft tube configuration is used to treat various industrial sludge. Here, the solid circulation rate is calculated for various sludge waste (having different density). This is reported in Table 3.

Table 3. Effect of sludge density on solid circulation rate

Sludge type	Sludge density (kg/m^3)	Solid circulation rate (kg/s)
Sewage sludge	3000	0.34
Paper sludge	2500	0.44
Activated sludge	1500	0.87

The solid circulation magnitude is observed to be highest when the activated sludge (having low density) is used.

Further, the effect of temperature on drying of activated sludge waste is calculated using CFD model for temperatures 320 K to 440 K for the initial moisture content of the sludge waste of 0.15 kg/kg. This is reported in Table 4. The amount of moisture removed from the sludge waste is comparatively larger when the inlet gas temperature is 400 K. The data points beyond 400 K are not reported (Table 4) because of unstable spouting behavior. Hence, 400 K is considered to be optimum for the drying of sludge waste.

Table 4. Effect of temperature on drying of activated sludge waste (initial moisture content = 0.15 kg/kg and mass = 3.91 kg)

Gas temperature (K)	Final mass of solid (kg)
320	3.71
340	3.62
360	3.56
380	3.48
400	3.41

Summary and conclusion

The flow pattern of gas-solid in a spouted bed column is investigated using Euler-Eulerian CFD model with KTGF. The calculated solid circulation rate is compared with conventional spouted bed column to analyze its performance. This is analyzed for various draft tube lengths, entrainment heights and draft tube diameters to find the optimum draft tube configuration that supports spouting behavior. The optimized draft tube configuration was found to be 20 cm (draft tube length), 4 cm (entrainment height) and 2.2 cm (draft tube diameter) for drying applications. The optimized system was used for drying of various industrial sludge waste. The rate of drying was found to be higher when the inlet gas temperature is high.

Thus, the CFD simulation helps to study the detailed gas-solid flow pattern in the spouted bed column and to find optimum conditions that support spouting behavior and drying rate.

References

1. C. R. F. Souza and W. P. Olivera, *Can. J. Chem. Eng.,* **87(2)**, 279.
2. W. Shuyan, S. Dan, L. Yikun and S. Wang, *Chem. Eng. Sci.*, 2010, **65**, 1322.
3. G. Lopez, A. Erkiaga, M. Amutio and M. Olazar, *Fuel,* 2015, **153**, 393.
4. R. K. Konduri, E. R. Altwicker and M. H. Morgan, *Can. J. Chem. Eng.,* 1995, **13(5)**, 744.
5. C. R. Duarte, M. Olazar, V. V. Murata and M. A. S. Barrozo, *Powder Technology*, 2009, **188(3)**, 195.
6. G. Szafran and A. Kmiec*, Indian Eng. Chem. Res.,* 2004, **43**, 1113.
7. X. L. Zhao, S. Q. Li, G. Q. Liu, Q. Song and Q. Yao, *Powder Technology*, 2008, **183(1)**, 79.
8. M. J. San Jose, S. Alvarez and F. J. Penas, *Chem. Eng. Sci.*, 2013, **100**, 413.
9. A. S. Souza and R. Battega, *Indian Eng. Chem. Res.,* 2018.
10. B. Ashraf Ali, Ch. Siva kumar and S. Pushpavanam, *Chemical Engineering Journal,* 2008, **144(3)**, 442.
11. B. Thorley, J. B. Saunby, K. B. Mathur and G. L. Osberg, *Can. J. Chem. Eng.*, 1959, **37**, 184.