PT-02

PREDICTION OF PARTICLE RESIDENCE TIME IN AN IMPINGING STREAM DRYER VIA CFD-DEM MODEL

*Patiwat KHOMWACHIRAKUL[1], Sakamon DEVAHASTIN[2], Thanit SWASDISEVI[3] and Somchart SOPONRONNARIT[1]

[1]*Division of Energy Technology, School of Energy, Environment and Materials, King Mongkut's University of Technology Thonburi, 126 Pracha u-tid Road, Tungkru, Bangkok 10140, Thailand*

[2]*Department of Food Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, 126 Pracha u-tid Road, Tungkru, Bangkok 10140, Thailand*

[3]*Division of Thermal Technology, School of Energy, Environment and Materials, King Mongkut's University of Technology Thonburi, 126 Pracha u-tid Road, Tungkru, Bangkok 10140, Thailand*

Corresponding author: Patiwat KHOMWACHIRAKUL. E-mail: pati_sdt@hotmail.com

ABSTRACT

Impinging stream dryer (ISD) is an excellent alternative to flash dryer for removing surface moisture from a particulate material. One of the most important pieces of information that is required for effective design of an ISD is the residence time of particles within the dryer. In this study, the mean residence time and residence time distribution (RTD) of particles within an ISD were simulated using a computational fluid dynamics (CFD) model as well as a combined computational fluid dynamics and discrete element method (CFD-DEM) model. The simulated results were compared with the available experimental data. The results showed that the mean residence time predicted via the CFD-DEM model was higher than that predicted via the CFD model; the predicted residence time via the CFD-DEM model was closer to the experimental data. CFD-DEM model predicted the RTD curves corresponding to the fact that more particles stayed longer in the system due to the effect of inter-particle collision.

Keywords: High-moisture particles; Pneumatic dryer; Simulation; Unhindered drying rate period

INTRODUCTION

Impinging stream dryer (ISD) is an alternative for drying high-moisture particulate materials. ISD involves the use of two or more streams that are introduced into a drying chamber at higher velocities in opposite directions; each stream may also carry a material to be dried [1]. Collision of the opposite streams causes turbulence and high shear rate, which in turn lead to enhanced rates of heat, mass and momentum transfer between the drying medium and particles, especially when drying is in the unhindered period. Since particles spend a very short time in the drying chamber of an ISD, it is very difficult to experimentally study the multiphase transport phenomena in this type of dryer. A number of investigators have therefore resorted to computa-tional tool to study the multiphase transport phenomena within an ISD and similar gas-particle contactors. For example, Choicharoen et al. [2] presented a mathematical model to simulate the flow and drying characteristics of an ISD at a steady state condition. Particle-wall interaction was considered in the simulation, but the effects of interactions between particles were neglected. The model could predict the experimental data in terms of the mean outlet particle moisture content, air humidity ratio and particle mean residence time only within ±10%. This obviously led to unrealistic simulated results, especially when particle motion behavior is of interest. An alternative method, namely, discrete element method (DEM), has become popular as a tool to simulate a flow field with dense particles where particle motion is affected by collis-ions. Recently, a number of researchers [3-8] have applied combined computational fluid dynamics (CFD) and DEM (CFD-DEM) to investigate the gas-solid flow behavior in a fluidized bed, spouted bed, cyclone separator and pneumatic conveyor. These investiga-tors noted that the calculated results on particle velocity profiles, particle mixing, minimum fluidizing velocity, minimum spouting velocity and bubbles formation agreed well with the experimental data. Despite its advantages, no application of CFD-DEM to an ISD has so far been made.

For this reason the aim of this work was to investigate the mean residence time as well as residence time distribution (RTD) of particles within an ISD using the CFD-DEM model. The simulated results were also compared to the CFD model and available experimental data.

MATHEMATICAL MODEL

Combined CFD and DEM model with soft-sphere model was used to investigate the residence time of particles in an ISD. The governing equations could be written as follows:

Equation of fluid motion
Continuity equation:

$$\frac{\partial}{\partial t}(\alpha\rho) + \frac{\partial}{\partial x_i}(\alpha\rho u_i) = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial t}(\alpha\rho u_j) + \frac{\partial}{\partial x_i}(\alpha\rho u_i u_j) = -\alpha\frac{\partial P}{\partial x_j}$$
$$+ \frac{\partial}{\partial x_i}\left[\alpha(\mu+\mu_t)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right] + \alpha\rho g + M_F \tag{2}$$

Turbulence model:

Realizable k-e model was used to repre-sent the gasphase turbulence. The realizable k-e equations can be expressed as [9]:

$$\frac{\partial(\alpha\rho k)}{\partial t} + \frac{\partial(\alpha\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\alpha\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]$$
$$+ \alpha G_k - \alpha\rho\varepsilon \tag{3}$$

$$\frac{\partial(\alpha\rho\varepsilon)}{\partial t} + \frac{\partial(\alpha\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\alpha\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]$$
$$- \alpha\rho\left(C_1 S\varepsilon - C_2\frac{\varepsilon^2}{k+\sqrt{\upsilon\varepsilon}}\right) \tag{4}$$

Equation of particle motion

The motion of individual particles was calculate by Newton's second law of motion, which includes the gravitational force ( ), $_gF$ gas-solid drag force ( ) $_DF$ and contract force between the particles ( ) $_cF$. In this work, the contact force was solved by DEM with a soft-sphere model [5, 10].

Newton's equation of motion is written as:

$$m_p\frac{du_p}{dt} = F_g + F_D + F_C \tag{5}$$

The drag force was calculated using the sum of the reactionary forces between gas and particles and the pressure gradient descri-bed in detail by Liu et al. [11]. Basically, the contact force is composed of normal and tangential components [5].

MODEL ASSUMPTIONS AND SIMULA-TION CONDITIONS

The following assumptions were adopted in the present study: air was an ideal gas; properties of air, including viscosity, thermal conductivity and heat capacity, were specified as a polynomial function of temperature [12]; particles (okara) had uniform spherical shape and did not undergo any change of shape during/after collision; rotation of particles during their movement was neglected.

The ISD chamber geometry of Choi-charoen et al. [2] was used for the simulation (see Fig. 1). The model equations were solved using the finite volume method. For the gas phase, the SIMPLE algorithm was used to treat the coupling between the pressure and velocity, while the spatial discretization for the conservation equations was taken care of via the power-law scheme. The utilized gasphase time step was 4 1 10′ - s. For the particle phase, the particle equa-tion was solved using an implicit discretization scheme. The time step of 6 1 10′ - s was used for the simulation of the particle motion. For the collision, the spring-dashpot force law was used with a spring constant of 1000 N/m. The conver-gence criterion was set at 3 10- for all equations. 227142 elements were noted to give grid-independent results and were used in this study. The properties of the particles and experimental data are described in detail by Choicharoen et al. [2]. The simulation conditions are presented in Table 1.
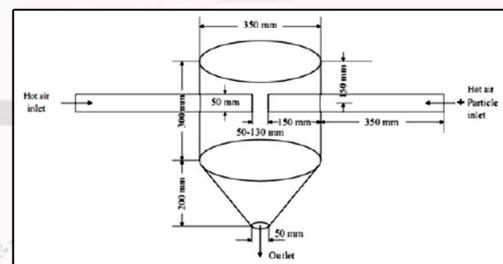

Fig.1   Schematic diagram of the simulated ISD [2].

Table 1   Conditions of simulation

Parameter used in simulation	CFDDEM model	CFD model	Experimental data [2]
Friction coefficient	0.3	-	-

Parameter used in simulation	CFDDEM model	CFD model	Experimental data [2]
Restitution coefficients			
Particle-particle	0.9	-	-
Particle-wall	0.9	0.9	-
Inlet air velocity (m/s)	20, 27	20, 27	20, 27
Particle mass flow rate (kg_dry solid/h)	10	10	10
Impinging distance (cm)	5, 13	5, 13	5, 13

BOUNDARY CONDITIONS

At the inlets inlet velocity and temperature boundary conditions were specified; flat velocity profile was assumed. The turbulence intensity was assumed to be 5% and the hydraulic diameter was used for any required calculations. The particle velocity was assumed to be zero at the inlet. At the wall no-slip boundary condition was assumed for the gas phase. The outlet was assumed to be at an atmospheric pressure.

RESULTS AND DISCUSSION

Fig.2(a) and (b) show a comparison between the simulated and experimental mean residence time of particles as a function of the inlet air velocity. The mean residence time of the particles increased with an increase in the inlet velocity, especially when the impinging distance was longer. This is because higher inlet velocity implies higher momentum of the particles to penetrate into the opposite stream and perform oscillatory motion within the impingement zone. This in turn led to the longer time the particles stayed in the system. The particle mean residence time obtained via the CFD-DEM model was longer than that obtained via the CFD model; the results from the CFD-DEM model agreed better with the experimental data. The particle mean residence time as predicted by the CFD-DEM and CFD models differed from the experimental results by ±4% and ±8%, respectively. The longer residence time obtained via the CFD-DEM model was due to the inclusion of the interparticle collision effect in the model. Collision of particles was hypothesized to produce more penetration of the particles into the opposing stream and prolong the motion of the particles in the impingement zone [2],

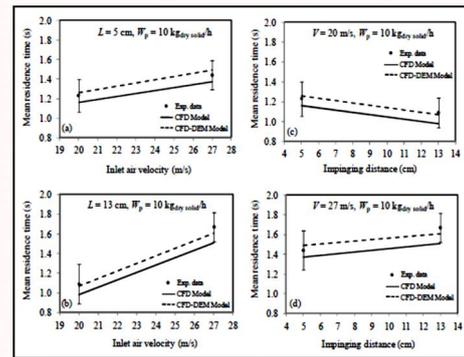resulting in increased residence time of the particles.

Fig.2 Comparison between simulated and experimen-tal mean residence time of particles within ISD (experimental data are those of Choicharoen et al. [2])

Fig 2(c) and (d) show a comparison between the simulated and experimental mean residence time of particles as a function of the impinging distance. The mean residence time of the particles decreased with an increase in the impinging distance when the inlet velocity was lower. This is because a lower air velocity led to a decrease in the momentum of the particles. Consequently, most particles released from the opening of the inlet pipe did not have adequate momen-tum to penetrate through the opposite stream and immediately escaped from the impinge-ment zone. This led to shorter mean residence time of the particles. On the other hand, the mean residence time of the particles increased with an increase in the longer impinging distance when the inlet velocity was higher. This is because the higher inlet velocity led to higher momentum of the particles to penetrate into the opposite stream. A longer distance in this case allowed the particles to undergo oscillatory motion within the impingement zone and hence stayed longer in the system [2]. The mean residence time obtained via the CFD-DEM model was again longer than that obtained via the CFD model.
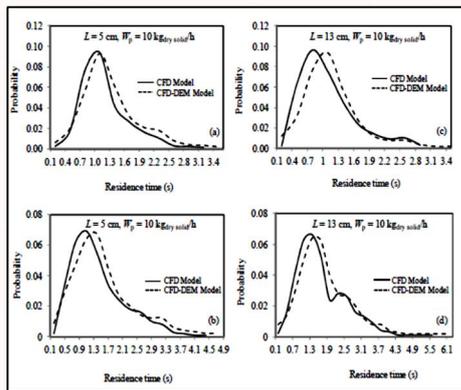
Fig.3 Simulated residence time distribution of particles. (a) and (c) inlet air velocity of 20 m/s; (b) and (d) inlet air velocity of 27 m/s.

Fig.3 shows a comparison between the simulated residence time distribution (RTD) of particles as obtained via the CFD-DEM and CFD models. The differences between the two model predictions can be seen through the distribution curves. The peak of the RTD curves shifted towards the right when the CFD-DEM model was

used to predict the RTD, indicating that more particles stayed longer in the system and hence longer mean residence time. The tailing areas (right side of the RTD curves) predicted by the CFD-DEM model were also larger than those predicted by the CFD model. This implies that the particles modeled by the CFD-DEM model stayed longer in the system, again due to the effect of inter-particle collision as mentioned earlier.

CONCLUSIONS

The mean residence time and RTD of particles within an ISD were predicted both by the CFD and CFD-DEM models. The simulated results showed that the mean residence time obtained via the CFD-DEM model was longer than that obtained via the CFD model. The longer resi-dence time was hypothesized to be due to the inclusion of the inter-particle collision effect in the CFD-DEM model. CFD-DEM model yielded the predictions that agree better with the experimental data; the mean residence time could be predicted within ± 4%. The peak of the RTD curves obtained via the CFD-DEM model shifted towards the right and the tailing areas predicting by the CFD-DEM model were also larger than those predicted by the CFD model, implying that the particles modeled by

the CFD-DEM model stayed longer in the system.

NOMENCLATURE

C_1, C_2 = Turbulence model constants
F_c = Contact force (N)
F_D = Drag force (N)
F_g = Gravitational force (N)
g = Gravitational acceleration (m/s²)
G_k = Production of turbulence kinetic energy due to mean velocity gradients
k = Turbulence kinetic energy (m²/s²)
L = Impinging distance (cm)
m_p = Mass of particle (kg)
M_F = Source term in momentum balance equation
S = Modulus of mean rate of strain tensor
t = Time (s)
u_i, u_j = Gas-phase velocity component (m/s)
u_p = Particle velocity (m/s)
W_P = Particle mass flow rate (kg_dry solid/h)
x_i, x_j = *Coordinates* (m)

GREEK LETTERS

α = Void fraction
ε = Energy dissipation rate (m²/s³)
μ = Gas dynamic viscosity (kg/m· s)
μ_t = Turbulent viscosity (kg/m· s)
υ = Kinematic viscosity (m²/s)
ρ = Gas density (kg/m³)
σ_k = Turbulent Prandtl number for k
σ_ε = Turbulent Prandtl number for ε

REFERENCES

[1] Tamir, A. Impinging-stream reactors: Fundamen-tals and applications. Elsevier, Amsterdam (1994)

[2] Choicharoen, K., Devahastin, S. and Soponron-narit, S. Numerical simulation of multiphase transport phenomena during impinging stream drying of a particulate material. Drying Technology, 30: 1227- 2012 (2012)

[3] Tsuji, Y., Kawaguchi, T. and Tanaka, T. Discrete particle simulation of two-dimensional fluidized bed. Powder Technology, 77: 79-87 (1993)

[4] Xu, B.H. and Yu, A.B. Numerical simulation of the gas-solid flow in a fluidized bed by combining discrete particle method with computational fluid dynamics. Chemical Engineering Science, 52: 2785- 2809 (1997)

[5] Sae-Heng, S., Swasdisevi, T. and Amornkitbam-rung, M. Investigation of temperature distribution and heat transfer in fluidized bed using a combined CFDDEM model. Drying Technology, 29: 697-708 (2011)

[6] Zhu, R.R., Zhu, W.B., Xing, L.C. and Sun, Q.Q. DEM simulation on particle mixing in dry and wet

particles spouted bed. Powder Technology, 210: 73- 81 (2011)

[7] 7. Chu, K.W., Wang, B., Xu, D.L., Chen, Y.X. and Yu, A.B. CFD-DEM simulation of gas-solid flow in a cyclone separator. Chemical Enginee-ring Science, 66: 834-847 (2011)

[8] Brosh, T. and Levy, A. Modeling of heat transfer in pneumatic conveyer using a combined DEM-CFD numerical code. Drying Technology, 28: 155-164 (2010)

[9] Wu, C., Cheng, Y., Ding, Y. and Jin, Y. CFD-DEM simulation of gas-solid reacting flows in fluid catalytic cracking (FCC) process. Chemical Engineering Science, 65: 542-549 (2010)

[10] Tsuji, Y., Tanaka, T. and Yonemura, S., 1998. Cluster patterns in circulating fluidized beds predicted by numerical simulation (discrete particle model versus two fluid model). Powder Technology, 95: 254-264 (1998)

[11] Liu, D., Bu, C. and Chen, X. Development and test of CFD–DEM model for complex geometry: A coupling algorithm for Fluent and DEM. Computers and Chemical Engineering, 58: 260-268 (2013)

[12] Frydman, A., Vasseur, J., Moureh, J., Sionneau, M. and Tharrault, P. Comparison of superheated steam and air operated spray dryers using computational fluid dynamics. Drying Technology, 16: 1305-1338 (1998)