Chemical Engineering Science 170 (2017) 365–377

Contents lists available at ScienceDirect

Chemical Engineering Science

journal homepage: www.elsevier.com/locate/ces

3D numerical simulation of a rotating packed bed with structured stainless steel wire mesh packing

Yi Liu, Yong Luo, Guang-Wen Chu *, Jiang-Zhou Luo, Moses Arowo, Jian-Feng Chen

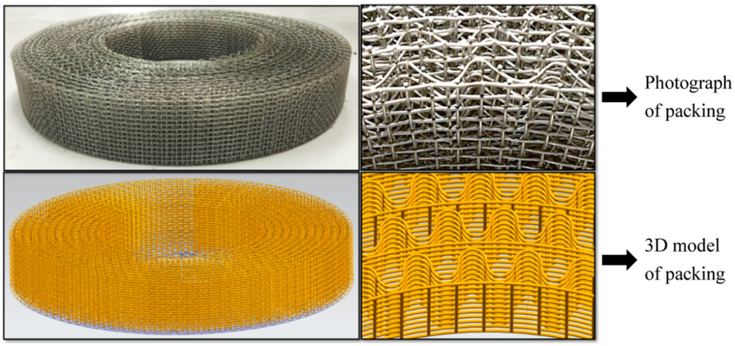
State Key Laboratory of Organic-Inorganic Composites, Beijing University of Chemical Technology, Beijing 100029, PR China
Research Center of the Ministry of Education for High Gravity Engineering and Technology, Beijing University of Chemical Technology, Beijing 100029, PR China

HIGHLIGHTS

- A 3D physical model of rotating packed bed with structured packing was built.
- CFD simulation results of gas pressure drop agreed well with experimental data.
- Gas pressure drop of the inner cavity zone is the major contributor to overall gas pressure drop.
- Detailed gas flow inside the RPB was obtained by the simulation results.
- Gas-side end effect at the outer annular packing zone was proposed.

GRAPHICAL ABSTRACT

The 3D packing's model, which has same structure and size as the physical packing, was employed for the CFD simulation of the gas pressure drop and gas flow in a RPB.



ARTICLE INFO

Article history:
Received 16 August 2016
Received in revised form 15 November 2016
Accepted 12 January 2017
Available online 14 January 2017

Keywords:
Rotating packed bed
3D model
CFD
Structured packing
Gas pressure drop

ABSTRACT

Computational fluid dynamics (CFD) is a powerful tool used to investigate the hydrodynamics in various chemical devices. As one of the typical process intensification equipment, rotating packed beds (RPBs) have been widely used in various fields. However, it is still a challenge to obtain the detailed information by CFD analysis due to the complex packing structure in the rotor of a RPB. In this study, we firstly built a three dimensional (3D) physical model with the same structure and size as the physical RPB and structured stainless steel wire mesh packing. The realizable $k-\varepsilon$ model is applied to investigate the gas pressure drop, pressure distribution, and gas flow at different rotational speeds and gas flow rates. Based on the breakdown of the overall gas pressure drop, the gas pressure drop in the inner cavity zone is the major contributor to the overall gas pressure drop under most operation conditions. The 3D physical model describing the actual RPB can give deep understanding of the gas flow in the entire RPB as well as the gas flow behavior around every fiber of the packing. Furthermore, one special phenomenon of strong turbulence, named as gas-side end effect, was revealed in the outer annular packing zone in the rotor.

---

* Corresponding author at: Research Center of the Ministry of Education for High Gravity Engineering and Technology, Beijing University of Chemical Technology, Beijing 100029, PR China.

E-mail address: chugw@mail.buct.edu.cn (G.-W. Chu).

http://dx.doi.org/10.1016/j.ces.2017.01.033
0009-2509/

Nomenclature

$C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ	constants
G	gas flow rate, m^3/h
N	rotational speed, r/min
k	turbulence kinetic energy, m^2/s^2
ΔP	pressure drop, Pa
P	pressure, Pa
U_k	velocity, m/s
x_k	position, m

Greek letters

ρ	density, kg/m^3
ε	turbulence energy dissipation rate
μ_t	viscosity, $mPa \cdot s$
σ_k	surface tension, N/m

1. Introduction

Computational fluid dynamics (CFD) is a useful tool for simulation and analysis of fluid flow, mass transfer, heat transfer, and chemical reaction (Versteeg and Malalasekera,1995). Several advantages of CFD simulation include: (1) ability to give more visible flow patterns inside reactors, (2) easy provision of detailed information, (3) accurate prediction of flow status prior to experiments, and (4) reducing cost, time, and risk associated with running repeated experiments (Kaya et al., 2014).

Chemical devices can be divided into two categories: static and rotational devices. The static devices analyzed by CFD include packed columns (Lautenschleger et al., 2015; Li et al., 2015a; Sebastia-Saez et al., 2015), tray columns (Malvin et al., 2014; Yang et al., 2015a,b; Zarei et al., 2013), fluidized beds (Shah et al., 2015; Zhang et al., 2015a,b,c), dryers (Keshani et al., 2015; Misha, 2014; Misha et al., 2013), heat exchangers (Al-Waked et al., 2015; Awan et al., 2015; Khoshvaght-Aliabadi et al., 2015), cyclone separators (Kuang et al., 2014; Sgrott et al., 2015; Zhang et al., 2015a,b,c) etc. Rotational devices such as stirred tank (Alfaro-Ayala et al., 2015; Eng and Rasmuson, 2015; Khapre and Munshi, 2015), blower (Baloni et al., 2015; Krishna et al., 2013; Wagh and Panchagade, 2014), and centrifugal pump (Ayad et al., 2015; Junaidi et al., 2015; Zhang et al., 2015a,b,c) have been also studied by CFD simulations. In general, the frame motion in steady state or mesh motion in transient state has been employed to simulate the single phase of gas flow in a blower or liquid flow in a pump.

The complex geometry of the internal components in some static equipment, such as the packing inside a packed column, hinders the rapid development of CFD simulation in such devices to some extent (Mahr and Mewes, 2007). Although the geometry of the internal components of rotating equipment is not as complex as that of the packing, the rotational motion notably increases the difficulty of CFD simulations.

A rotating packed bed (RPB), known as Higee (high gravity) device, was invented by Ramshaw (Ramshaw and Mallinson, 1981). Under the centrifugal field, the mass transfer and micromixing of a RPB can be up to 1–3 orders of magnitude larger than that of a conventional packed column, exhibiting prominent process intensification characteristics (Chen, 2003; Zhao et al., 2010). Consequently, RPBs have been successfully applied to absorption (Guo et al., 2014; Li et al., 2015b; Sun et al., 2015; Zhang et al., 2013a,b), distillation (Chu et al., 2013), ozone oxidation (Zeng et al., 2013), vacuum deaeration (Yang et al., 2016), etc. Though the RPB has the complex packing structure as the packed column and the rotational motion as the rotational equipment, there are some progresses of the RPB CFD simulations and Table 1 gives a summary of CFD simulation studies of Higee devices.

Xu (2004) used SIMPLE arithmetic to study the liquid flow in a simplified RPB model. The Boussinesq assumption was imposed on the $k-\varepsilon$ equations in order to obtain the dispersed liquid flow and the mass transfer coefficient of the liquid phase. Llerena-Chavez and Larachi (2009) employed the porous medium model to describe the packing inside a RPB, and built a three dimensional (3D) physical RPB model. The effect of gas feed entrance on the gas maldistribution was investigated numerically and then experimentally validated. Yang et al. (2010) developed both two dimensional (2D) and 3D models to predict the velocity of single gas phase flow at different rotating speeds, and the simulation results show that the models used to describe the RPB can give rise to an understanding of the gas flow in a RPB. Martínez et al. (2012) simulated multiphase flow of water-SO_2 in a RPB. Shi et al. (2013) employed the Yang's geometrical model and developed a 2D CFD computational model to study the multiphase flow RPBs. Yang et al. (2015a,b) suggested optimum structures in the cavity zone of the RPB based on the results of CFD simulations. Recently, the gas-liquid mass transfer (Yang et al., 2016) and micromixing processes (Guo et al., 2016) are performed in RPBs via CFD simulations. To sum up, there are generally two kinds of CFD simulation for Higee devices: (1) 3D CFD simulation: a porous media model was adopted. This approach adds a momentum sink in the governing momentum equations (ANSYS Fluent User's Guide, 2013) and describes the packing by a mathematic model; (2) 2D CFD simulation: the cylindrical packing fibers were simplified into 2D squares. This square model describes the packing structure on x- and y-coordinates.

Due to the simplification of the real packing geometry in the simulation, it is difficult to obtain the detailed information of fluid flow around fibers of wire mesh packing utilizing the above men-

Table 1
A summary of RPB CFD simulations.

References	Packing	Phase	Mesh number	Contents
Xu (2004)	Mathematic model	Gas	–	2D & 3D gas flow in RPB
Llerena-Chavez and Larachi (2009)	Porous media model	Gas	350,000	3D gas flow and pressures drop
Yang et al. (2010)	Porous media model	Liquid	1,000,000	2D & 3D micro-mixture
Martínez et al. (2012)	Porous media model	Gas & liquid	2,270,000	H_2O-SO_2 system in RPB
Shi et al. (2013)	2D small squares	Gas & liquid	1,420,000	2D gas & liquid Flow in RPB
Yang et al. (2015a,b)	Porous media model	Gas	3,062,309	3D gas Flow
Yang et al. (2016)	2D small squares	Gas & liquid	660,601	2D vacuum deaeration
Guo et al. (2016)	2D small squares	Liquid	1,420,000	2D micromixing