Chemical Engineering Journal 329 (2017) 66–76

Contents lists available at ScienceDirect

Chemical Engineering Journal

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

Investigation of behaviors of the circulating fluidized bed

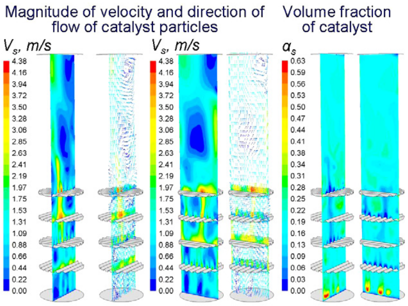
O.P. Klenov [a],*, A.S. Noskov [a], O.A. Parahin [b]

[a] *Boreskov Institute of Catalysis, Siberian Branch of the Russian Academy of Sciences, Pr. Akad. Lavrentieva 5, 630090 Novosibirsk, Russia*
[b] *JSC "NPK Sintez", Barnaul, Russia*

HIGHLIGHTS

- The circulating fluidized bed with internal horizontal baffles has been investigated.
- Eulerian–Eulerian approach with Gidaspow gas-solid drag model and standard k-e model.
- Mass of catalyst in the empty space between the baffles and in the baffles area.
- Evaluation of residence time for catalyst particle in the column versus flow rate.

GRAPHICAL ABSTRACT



ARTICLE INFO

Article history:
Available online 19 June 2017

Keywords:
Circulating fluidized beds
Gas-solid flow
Fine microspheric particles of catalyst
Pressure measurements
CFD simulation
Eulerian–Eulerian approach

ABSTRACT

The circulating fluidized bed, separated by horizontal baffles, was studied both experimentally and numerically by CFD methods. The pressure distribution in a large-scale cold-flow column with a diameter of 700 mm with a circulating fluidized bed of a microspheric catalyst was measured experimentally for the superficial velocity range from 0.05 to 1.0 m/s. The column was sectioned in height by four rows of horizontal baffles made of several angle bars. For this column design 3D CFD simulations were performed based on the Eulerian–Eulerian approach (two-fluid model) with standard closures and using a coarse computational grid.

Comparison of the results of the experiment and transient simulations showed that simulation allows estimating the pressure distribution in a fluidized gas-solid apparatus with an error not exceeding 5%. This prediction accuracy can be sufficient for important practical applications, for example, for the calculation and design of the apparatus or the choice of the design of distributive baffles.

The pressure distribution over the height of the column is largely determined by the distribution of the catalyst particles inside column. For example, the contribution of the particle weight to the static pressure is approximately 96% for the superficial velocity Wo ~ 0.1 m/s. Therefore, this kind of simulation can provide useful information for the distribution of solid particles in a fluidized bed apparatus. This is important for a large number of users of commercial CFD packages, which are engaged in the design of devices that provide optimal hydrodynamic conditions.

1. Introduction

Large-scale devices with a circulating fluidized bed are of great interest for industrial applications [1,2]. Development and design of efficient and safe apparatus should be performed on the basis of sufficiently accurate predictions of the behavior of the gas-particle flow bed in wide intervals of process parameters.

At present, simulation of the fluidized bed in industrial reactors has encountered a number of difficulties, where one of the main is the extremely large number of small particles and a correspondingly very large ratio of the spatial dimensions of the apparatus

---

* Corresponding author.
E-mail address: klen@catalysis.ru (O.P. Klenov).

http://dx.doi.org/10.1016/j.cej.2017.06.092
1385-8947/

Nomenclature

C_D	Drag coefficient
d	diameter, m
e	restitution coefficient of particle collision
G	generation of turbulence kinetic energy caused by the mean velocity gradients
g	radial distributional function, gravitational acceleration, m/s^2
h	height, m. Average mesh size, mm
I	unit tensor
k	turbulent kinetic energy per unit mass, J/kg
L	length scale, m
M	mass of catalyst, kg
N	number of control volumes
p	static pressure, Pa. Formal order of accuracy of simulation
r_{ij}	ratio of average mesh size h_i and h_j
Re	Reynolds number
t	time, s
$\vec{v}$	overall velocity vector, m/s
V	velocity magnitude, m/s
V_0	total volume of model, m^3
W_o	inlet velocity of fluidized air, m/s

Greek letters

α	volume fraction
β	interactional momentum exchange coefficient
ε	turbulent dissipation rate, m^2/s^3
η	viscosity ratio
Θ	granular temperature
μ	viscosity, kg/(m*s)
ξ	bulk viscosity, kg/(m*s)
ρ	density, kg/m^3
σ	dimensionless parameter
τ	stress tensor, Pa
φ	time-averaged static pressure, Pa

and the individual particle. This makes it impossible to use the most accurate computational method – Eulerian-Lagrangian treatment for simulating the entire fluidized bed apparatus. These limitations are related to the capabilities of modern computer technology and the time spent on performing transient simulations.

The use of the Eulerian–Eulerian CFD model (the two-fluid model) makes it possible to carry out this kind of simulating, but the computational capabilities also impose restrictions that lead to the use of coarse computed grids. The size of the computed grid cells, as a rule, exceeds by at least two orders of magnitude the characteristic particle size of the solid phase.

It has been shown in the literature that the use of coarse grids leads to the fact that simulation results can lose such important effects as the effect of unstable meso-scale clusters of particles that have a spatial scale smaller than the size of the computational grid [3,4]. Various modes are known to involve these effects in the simulation process, for example, using suitable subnet models [5,6]. Nevertheless, the verified and validated methods of refinement of the two-fluid model, recognized by the scientific community, are absent by now [7].

When designing a circulating fluidized bed apparatus, it is not necessary to know the entire amount of accurate information that will received by CFD simulation. First of all, it is necessary to have data on the distribution of pressure along the height of the apparatus and the associated distribution of the amount of solid particles. The use of CFD simulation on a coarse grid within the two-fluid model and the standard closures of the hydrodynamic equations used in commercial CFD packages make it possible to calculate the above values for large-scale fluid bed installations. The question of whether the accuracy of such modeling is sufficient for practical applications related to the design of apparatus remains open.

Experimental data are available on the effect of certain types of baffles on hydrodynamics and heat-mass transfer in a fluidized gas-solid bed in the literature [8–15]. This mainly applies to perforated and screen baffles [9–12], as well as louver baffles [13,14]. One work is known where baffles of angles have been investigated [15]. Here tracer gas method was applied to study the mixing of gas and solids in a cold-model fluid coker stripper with several rows of baffles of angles. It has been shown that the design of baffles affects the efficiency of fluid coker stripper, as well as the circulation rate of solid particles. Stripping efficiency increased with increasing superficial stripping gas velocity at a constant solids circulation rate. Eight rows of baffles with 90°-top-included-angles provide a lightly larger stripping efficiency than six 90° top angle baffles rows and one 30° top angle baffle in the top row.

In the literature, there are a bit works on simulating fluidized layers with partitioning baffles. The effect of annular baffles was investigated in [16–18], where it was shown that the configuration of the annular deflector has a significant effect on the mixing of gas with a solid and the average distribution of solid particles in the riser. The effect of perforated baffles on the distribution of solid particles in the reactor was studied at relatively low velocities of the fluidizing gas velocity from 0.03 to 0.072 m/s [19].

Studies of the distribution of the average pressure over the height of the reactor with sets of baffles of angles, as well as the distribution of the solid particles over the height of column were not presented in the literature.

In the present paper, the results of experiments on a large-scale stand-up with a diameter of 0.7 m were compared with the results of numerical simulation within the framework of the two-fluid model. The calculations were performed on a coarse grid for which the ratio of the average grid size to the particle diameter was 155. The object of the study was a fluidized gas-solid bed that was sectioned by horizontal baffles. In the reactor model, four rows of distribution baffles were installed, made of steel angle bar (90° included-top angle) with a size of 50 × 50 mm. Sectioning of the fluidized bed with such sets of horizontal baffles is used in a number of industrial large-scale reactors, for example, for the dehydrogenation of C_3-C_5 paraffins to olefins. In experiments and simulations, spherical particles of class A by Geldart were used.

Two tasks were set. Firstly, on the basis of a comparison with the experimental data, it needs to estimate the possibility of using fluidized gas-solidification simulations in the framework of the Eulerian–Eulerian approach using a coarse computational grid. And second task was to estimate the distribution of pressure and solid particles in a reactor with a gas-solid, sectionalized fluidized bed.

2. Experimental set-up

Experimental studies have been performed on a large-scale model of the reactor with a fluidized bed. All the experiments have been conducted at room conditions with air as the fluidizing gas. A