Chemical Engineering Science 102 (2013) 335–345

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Experimental and numerical study of two-phase flows in arrays of cylinders

Pierre Horgue[a,c,*], Frédéric Augier[c], Paul Duru[a], Marc Prat[a,b], Michel Quintard[a,b]

[a] Université de Toulouse, INPT, UPS, IMFT (Institut de Mécaniques des Fluides de Toulouse), Allée Camille Soula, F-31400 Toulouse, France
[b] CNRS, IMFT, F-31400 Toulouse, France
[c] IFP Energies Nouvelles, rond-point de l'échangeur de Solaize, BP 3, 69360 Solaize, France

HIGHLIGHTS

- Two-phase flow regimes in arrays of cylinders are characterized experimentally.
- A 2D approach is proposed to simulate the flow phenomenology previously observed.
- The 2D model includes terms taking into account the 3D effects of the experiment.
- 2D numerical simulations show good agreement with the experimental visualizations
- Qualitative study of gas injection is performed using the developed numerical method.

ARTICLE INFO

Article history:
Received 18 March 2013
Received in revised form
11 July 2013
Accepted 16 August 2013
Available online 24 August 2013

Keywords:
Multiphase flow
CFD
Volume-Of-Fluid
Packed bed
Pulsing flow

ABSTRACT

In this paper we study the spreading of a liquid jet in a periodic array of cylinders with a characteristic size of the passages between solid obstacles equal to 1.5 mm, close to the capillary length. An important outcome of our study is to show that this configuration allows most of the two-phase flow regimes described in the literature about trickle beds to be observed, even with no gas injection. Different aspects of the flow phenomenology have been studied, such as bubble creation and transport. As direct numerical methods for tracking interfaces would require too much computation time, especially in three-dimensional cases, we propose to simulate the two-phase flows observed experimentally with two-dimensional simulations corresponding to the spreading of a liquid jet in an array of disks. We show that this numerical approach allows the phenomenology observed experimentally to be reproduced satisfactorily. Hence, numerical simulations can be used subsequently to study the effects of specific parameters without setting up a new experimental procedure. As an example, the stabilizing effect of gas injection on the flow pattern is studied numerically.

1. Introduction

Gas–liquid downward flows are common in chemical reactors (Lee and Pang Tsui, 1999). When a stationary solid phase is involved, different hydrodynamic regimes can occur depending on the flow rate and properties of each phase. This is the case in fixed bed catalytic reactors (Larachi et al., 1991) commonly used in the refining industry for the hydroprocessing of petroleum cuts. In such reactors, spherical or extrudate catalyst particles of 1–4 mm of diameters are generally used. Following previous studies (Charpentier and Favier, 1975; Ng and Chu, 1987), 4 different regimes are identified, but the most common regime is the trickling one. Despite simple flow characteristics in Trickle Bed Reactors (TBRs), hydrodynamic behavior of such reactors has been the object of many studies (Al-Dahhan et al., 1997). This is partly due to the high sensitivity of local flow patterns to physical properties of the fluids and solid phases, in addition to interface properties as contact angles and surface tensions (Baussaron et al., 2007; Julcour-Lebigue et al., 2009). In particular, exact position of transitions between well characterized regimes as trickling, bubbly or pulsed regimes are not exactly known. Moreover, physical mechanisms responsible of regime modifications and transitions are not completely understood and modeled. Maldistributions in TBRs are known to be very difficult to avoid completely while being responsible of important losses of reaction efficiency (Strasser, 2010). They can be generated by poor homogeneity in either the loading of catalyst particles, the distributors at the top of

* Corresponding author at: Université de Toulouse, INPT, UPS, IMFT (Institut de Mécaniques des Fluides de Toulouse), Allée Camille Soula, F-31400 Toulouse, France. Tel.: +33 534 322 855.
E-mail address: pierre.horgue@imft.fr (P. Horgue).

0009-2509/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ces.2013.08.031

reactors, or both. In either case, complex flow redistribution mechanisms are involved. Non-invasive experimental techniques have been successfully applied to investigate TBRs, such as $\gamma-$tomography (Reinecke and Mewes, 1997; Schubert et al., 2008), ECT (Matusiak et al., 2010) or collecting devices at the bottom of columns (Marcandelli et al., 2000), but they are generally limited to analysis scales larger than catalyst particle length.

Pursuing the main objective of modeling flows inside TBRs, a first step consists in the study of the trickling flow in a quasi 2-dimensional configuration of an array of cylinders. The flow generated inside this configuration is appropriate to observe transition mechanisms that are also present inside complex 3-dimensional flows, like bridges between trickling liquid films. As the 2D setup is easy to investigate using optical techniques, this presents a good case to validate numerical modeling approaches that may be further used to simulate downward gas/liquid flows in 3-dimensional fixed bed of solid particles. The chosen configuration is reminiscent of the case of tube bundles and it is therefore interesting to examine the literature on this subject.

Two-phase flows in tube bundles, from microchemical reactors to heat exchangers, have been extensively studied because of the many industrial processes that are based on this geometry. Each process features specificities depending on the operating conditions such as the pressure, the temperature or the characteristic size of the system. In microfluidic devices, with a much smaller gap between solid obstacles than the capillary length, the capillary forces are dominant when compared to the gravity force. On the contrary, if we consider a heat exchanger composed of a tube bundle with a characteristic gap of about 1 cm, the capillary forces may be neglected compared to the gravitational force and often compared to inertial effects as well. The flow regimes, their properties, and the transitions between them, have been the purpose of many studies (Kondo and Nakajima, 1980; Ulbrich and Mewes, 1994; Xu et al., 1998; Noghrehkar et al., 1999) but remain poorly understood in the case of tube bundles. The major objective of any modeling attempt is to determine pressure drops and phase fractions which are intimately associated with the observed flow pattern.

The characterization of the flow patterns and their transitions has been studied experimentally by previous authors for different flow conditions, upward or downward, and for different tube arrangements, in-line or staggered. Flow regime maps have been plotted first for upward flow in staggered tube bundles (Kondo and Nakajima, 1980), then for an in-line arrangement (Ulbrich and Mewes, 1994). Xu et al. (1998) studied up and down-flow in an in-line tube bundle and Noghrehkar et al. (1999) studied the effect of the arrangement by performing experiments on both in-line and staggered tube bundles. Melli et al. (1990) draw flow regime maps in "two-dimensional" packed beds composed of a staggered arrangement of O-ring solids placed between two plates. The objective of that study was to simultaneously observe regime transition at the microscale, i.e. the scale of the passage between two solids, and in the O-ring bundle at the macro-scale,. More recently, Krishnamurthy and Peles (2007) studied the flow patterns for two-phase flows through a staggered bank of micro-pillars and highlighted the differences with previous studies at a larger scale. We must note that the flow maps, plotted in different experimental conditions and at different scales, have the same trends. However, phenomena related to flow regime changes at the scale of a tube bundle remain too poorly understood to develop general regime change laws.

Due to the various flow patterns that can exist in tube bundles, the influences of which are not fully assessed, modeling approaches to predict pressure drop and void fraction are generally empirical. Most of them are based on an adaptation of existing models, for instance in-tube two-phase flow models. The most commonly used approach is based on the Lockhart–Martinelli model (Lockhart and Martinelli, 1949) which relates the two-phase pressure drop to the one-phase pressure drop, through specific correlations. This approach was first developed for two-phase flow in pipes and then used for more complex geometries, including tube bundles. Such models have been used successfully for vertical up and down flows across horizontal tube bundles (Xu et al., 1998) and for gas–liquid flows across a bank of micropillars (Krishnamurthy and Peles, 2007). The effect of the pitch-to-diameter ratio (the pitch is the distance between the centers of two neighboring tubes in the tube bundle) was also studied for staggered and in-line tube bundles (Dowlati et al., 1992). Recently, Bamardouf and McNeil (2009) performed a comparison between various correlations available in the literature and found that the most accurate prediction in terms of void fraction is given by Feenstra et al. (2000) correlation and that Ishihara et al. (1980) correlation is the best way to predict the pressure drop.

Some numerical studies concerning the flow modeling at the pore scale, i.e. the scale of the passage between tubes, have been performed in an effort to understand more accurately the origin and transitions of the different regimes. The two-phase flow modeling at low Reynolds numbers, in which interfaces play an important role and need to be explicitly tracked, is a complex case due to numerical issues. Direct numerical methods for tracking interfaces, such as the "Volume-of-Fluid" method (Hirt and Nichols, 1981), have been successfully used to simulate two-phase flow in tubes (Gupta et al., 2009). However, even with a significant reduction of the computation time due to the axial symmetry of the problem, simulations still requires significant computation time. More recently, the Volume-Of-Fluid method has been validated in a three-dimensional case, the trickling flow on a stack of a few particles (Augier et al., 2010). However, this was possible for a small number of "grains" only and required long computation times (around one week on 2 processors for 1 s of physical time). The numerical simulation of two-phase flows at the scale of a tube bundle requires too much computation time under these conditions, which explains the lack of numerical simulations.

The experimental configuration used in this work is first presented in Section 2.1. It is based on the use of micromodels consisting in arrays of cylinders maintained within a Hele Shaw cell. The characteristic lengths of the system (cell thickness and minimum spacing between neighboring cylinders) are 1.5 mm, close to the capillary length for most fluids, which allows for an interesting competition between capillary, gravity, and inertial effects. Considering that the experimental device can be seen as a two-dimensional medium, two-dimensional numerical simulations on array of disks were performed using a numerical interface tracking method, namely the "Volume-Of-Fluid" method, implemented in the OpenFOAM® software, as described in Section 2.2. As already mentioned, direct numerical simulations are very time consuming, particularly in three-dimensions, which prevent the possibility to simulate the two-phase flow in the real geometry of the experiment with reasonable computation times. The constitutive equations of the present 2D numerical model have therefore been modified by adding terms which account for the three-dimensional effects present in the experimental device using simplifying assumptions. The visualization of spreading of a liquid jet injected in the micromodels led to the observation of different flow regimes, similar to those usually observed in trickle beds or in tube bundle flows, which were dependent on the injection flow rate. Both regime transitions and flow patterns properties are described in Section 3. The numerical results presented in Section 4 compare favorably with the experimental results, which confirm the ability to simulate the experimental phenomenology with a