chemical engineering research and design xxx (2014) xxx–xxx

Contents lists available at ScienceDirect

Chemical Engineering Research and Design

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

# CFD and experimental investigation of the gas–liquid flow in the distributor of a compact heat exchanger

*Selma Ben Saad*[a,b], *Caroline Gentric*[c,*], *Jean-François Fourmigué*[a], *Patrice Clément*[a], *Jean-Pierre Leclerc*[b]

[a] *CEA, DRT/LITEN/DTS/LETh-GRETh, 17 Rue des Martyrs, 38054 Grenoble Cedex 9, France*
[b] *Laboratoire Réactions et Génie des Procédés (LRGP), CNRS, UMR 7274, 1, rue Grandville BP 20451, 54001 Nancy, France*
[c] *Laboratoire Génie des Procédés Environnement Agroalimentaire (GEPEA), UMR CNRS 6144, 37 boulevard de l'Université, 44602 Saint-Nazaire Cedex, France*

ABSTRACT

High performance of compact heat exchangers is conditioned by correct fluid distribution. This is especially true for gas–liquid heat exchangers where a uniform distribution is particularly delicate to obtain and where maldistribution entails significant performance deterioration. Several phenomena can lead to phase distribution problems: the fins may be subject to manufacturing defects or fouling, leading to shortcuts or dead zones. But the first source of maldistribution may be a poor distribution at the outlet of the entrance distributor. This distributor aims at mixing the phases and distributing them across the channels.

The present study deals with the simulation and experimental investigation of the two-phase distribution and flow regimes in a distributor located at the bottom of the cold flow pilot plant of a vertical compact heat exchanger. Air and water are the working fluids, and the range of superficial velocities inside the distributor is 0.9–8.8 m $s^{-1}$ and 0.35–0.8 m $s^{-1}$, for air and water respectively. Three-dimensional Volume Of Fluid (VOF) simulations are performed and compared to experimental distributions, pressure drops, and visualizations.



*Keywords:* CFD simulation; Volume Of Fluid; Gas–liquid distribution; Compact heat exchanger; Flow regimes; Gas–liquid flow

## 1. Introduction

Maldistribution problems can be critical for the performance of two-phase compact heat exchangers. Fluid distribution inside the heat exchanger is first conditioned by the flow regimes inside the distributor and the gas–liquid distribution at its outlet. Moreover the two-phase flow regime affects the pressure drop, the stability of the system as well as the momentum, heat and mass transfer. Therefore, a better knowledge and a prediction of the flow regimes at the distributor outlet present a major interest for the design and operation of compact heat exchangers.

For single-phase manifold distribution systems, the common design rules for uniform distribution into parallel channels are to ensure that the pressure drop over the channels is much higher than the pressure drop across the header. In two-phase flows, many parameters affect the two-phase distribution: both design (distributor geometry, inlet configuration) and operating factors (flow rates, flow regimes, fluid properties) have an impact (Marchitto et al., 2008, 2009; Kim et al., 2011, 2012). Therefore the prediction of the hydrodynamics is particularly delicate.

Two-phase flow can lead to several configurations referred to as flow patterns or flow regimes; they affect pressure drop, heat and mass transfer characteristics. Predicting these flow patterns is thus crucial in an engineering point of view. Flow pattern maps or models have been established for common situations such as gas–liquid flows inside tubes with different

* *Corresponding author*. Tel.: +33 2 40 17 26 32.
E-mail address: caroline.gentric@univ-nantes.fr (C. Gentric).

Received 17 July 2013; Received in revised form 18 January 2014; Accepted 3 February 2014
0263-8762/$ – see front matter 
http://dx.doi.org/10.1016/j.cherd.2014.02.002

inclinations (Baker, 1954; Petalas and Aziz, 1998), in mini- or micro-channels (Triplett et al., 1999; Chung and Kawaji, 2004; Ide et al., 2007; Shao et al., 2009), but they cannot be used to predict the two-phase flow in complex geometries such as those of the distributing systems at the inlet of compact heat exchangers. The aim of such distributing systems is to mix as uniformly as possible the gas and liquid phases. It is thus necessary to perform experimental investigations to test new designs or new operating conditions. With the increase of computing resources, CFD can also be of help to predict two-phase flow configurations in such situations.

To mix gas and liquids before entering mini- or micro-channels, geometries such as T or Y junctions are now commonly employed. They were first studied experimentally: Garstecki et al. (2006) studied experimentally the mechanism of bubble formation in a microfluidic T-junction, they put into evidence that at small capillary numbers, the mechanism of formation of bubbles is not governed by shear stress but by the pressure drop across the emerging bubble which almost fills the channel cross-section; thus, this regime is typical of micro-systems. The study of Garstecki et al. was completed by detailed velocity field investigation using μPIV (van Steijn et al., 2007). Yue et al. (2008) investigated the flow patterns in Y-type microchannels: they observed bubbly, slug, slug-annular, churn and annular flow. Arias et al. (2009, 2010) analyzed experimentally the bubble generation by a T-junction composed of two 1 mm internal diameter capillaries. In these mini-channels, the mechanism of formation depends on a competition between capillary forces (which dominate over buoyancy and inertial forces) and drag due to the liquid cross-flow.

More and more, CFD direct simulation via interface tracking methods is used for a better understanding or a prediction of two-phase flow regimes or two-phase mixing. These numerical methods allow to determine the interface topologies. De Schepper et al. (2008) predicted two-phase flow regimes for gas–liquid co-current flow in horizontal ducts. Qian and Lawal (2006) simulated the formation of bubbles governed by the squeezing regime in a T-junction: they investigated the effects of surface tension, shear stress and pressure. Kashid et al. (2007) simulated the slug formation inside a 120° Y-junction. De Menech et al. (2008) characterized numerically the squeezing regime, the dripping regime, which is governed by shear stress, and the jetting regime, obtained in a microfluidic T-junction. Guo and Chen (2009) used the Volume Of Fluid (VOF) method to simulate the generation and the development of the Taylor bubbles at various operating conditions and fluid properties inside a micro-channel T-junction for a better understanding of Taylor bubbles' formation. Abadie et al. (2012) studied the effect of fluid properties and operating conditions on the generation of gas–liquid Taylor flow in microchannels experimentally and numerically. They studied bubble and slug lengths, liquid film hold-up and bubble velocities. Their results confirmed the model of Garstecki et al. (2006).

As far as mini-channels are concerned, Taha and Cui (2004, 2006) simulated slug flow inside 2 mm vertical capillaries with both square and circular cross-sections with the VOF method. Arias et al. (2012) performed direct numerical simulations of the bubble generation in a T-junction: they reproduced the bubble and slug flow patterns, as well as the bubble velocity, formation frequency and size observed experimentally. They observed bubble, slug, churn and annular regimes. When observing the bubble formation frequency as a function of the gas superficial velocity, they put into evidence two domains: at low $V_{SG}$ values, the bubble frequency evolves linearly with $V_{SG}$, while a saturation regime appears at higher $V_{SG}$ values.

The objective of this paper is to investigate experimentally and numerically the gas–liquid distribution and flow regimes inside a distributor located at the bottom of a cold-flow experimental test rig. Air and water are the considered gas and liquid phases respectively. They are mixed in Y-junctions inside the distributor. The influence of the fluid flow rates has been investigated. Experiments consist of single-phase flow distribution and two-phase flow pressure drop measurements as well as flow visualization using a high-speed camera to characterize the different flow regimes. To predict the gas–liquid hydrodynamics in the distributor, CFD simulations have been performed using the Volume Of Fluid (VOF) approach. In fact, this approach is suitable to describe gas–liquid interfaces with large surface topology changes at reasonable computational cost. Numerically predicted flow regimes, discrete phase size, and distribution of phases have been compared to experimental observations. In a first part, the fluid flow in the whole distributor geometry has been calculated, and in a second step a single channel has been considered to calculate more accurately the gas–liquid interface, and to assess the influence of the gas and liquid superficial velocities on the flow regime inside the distributor.

## 2. Experimental set-up and measurements

### 2.1. *Experimental test rig*

A description of the experimental test rig is presented in Fig. 1a (see also Ben Saad et al., 2011). The test section is a quasi two-dimensional vertical compact heat exchanger (height 1 m, width 1 m, thickness 7.13 mm) and consists of offset strip fins placed between two transparent flat plates. Fluids (gas and liquid) injections are designed to obtain two possible inlet flow configurations: either gas and liquid enter the distributor on the same side, or on each side. In the industrial practice, the inlet configuration: co-current or counter-current inlets of phases, depends on the fluid main positions relative to the heat exchanger header. In the present study, fluids (gas and liquid) injections are designed to obtain counter-current flows in the distributor (Fig. 1b). This horizontal distributor is rectangular and is described in Section 2.2. Its role is to mix phases before injection into the offset strip fin channels. Then, the mixture flows upwards through the test section.

At the top of the test section, the flow is divided into seven zones regularly distributed on the width. In each zone, a compartment allows separating gas and liquid by gravity and the flow rate of each phase in each zone is measured using gas and liquid flow meters to appreciate the uniformity of the phase distribution across the exchanger (thermal mass flow meters are used for the measurement of the air flow rates, Coriolis mass flow meters are used for the measurement of the water flow rates).

Two absolute pressure transducers are installed to measure the inlet pressures for gas and liquid. Differential pressure transducers are installed along four horizontal lines to measure pressure drop across the test section at different heights. In particular, the first line of measurement allows to measure the pressure drop across the distributor, which is of interest here.