EXPERIMENTAL ANALYSIS AND COMPUTATIONAL MODELLING OF GAS–LIQUID STIRRED VESSELS

G. Montante*, **A. Paglianti and F. Magelli**

Department of Chemical, Mining and Environmental Engineering, University of Bologna, Bologna, Italy.

Abstract: The aim of this work is to investigate the turbulent hydrodynamics of a gas–liquid stirred tank of standard geometry through experiments and simulations. The 2-D velocity fields are obtained by a two-phase PIV technique, consisting of a pulsed Nd:YAG laser, emitting light at 532 nm, and two cameras, each provided with a filter, that allow to discriminate between the light scattered by the fluorescent liquid seeding particles and that scattered by the bubbles. The experimental results obtained at different gas flow rates are presented, compared with single-phase data and discussed for gaining insight into the gas–liquid flows. They are also adopted for the quantitative evaluation of the results produced by CFD simulations based on a Two Fluid Model approach. The agreement between the experimental and the calculated mean velocity fields indicates that the selected CFD modelling is appropriate for the prediction of the mean hydrodynamic features of gas–liquid dispersions in stirred vessels.

Keywords: gas–liquid systems; two-phase PIV technique; CFD simulation; multi fluid model; liquid flow field; bubble velocity.

*Correspondence to:
Dr G. Montante, Department of Chemical, Mining and Environmental Engineering, University of Bologna, Viale Risorgimento 2, I-40136 Bologna, Italy.
E-mail: giusi.montante@mail.ing.unibo.it

DOI: 10.1205/cherd06141

0263–8762/07/
$30.00 + 0.00

Chemical Engineering Research and Design

Trans IChemE,
Part A, May 2007

INTRODUCTION

Gas–liquid stirred vessels have been widely investigated over the past decades, but, due to the complex nature of phase interaction, a detailed and quantitative description of their fluid dynamic behaviour and the determination of the two-phase hydrodynamic variables is still a particularly complicated task. Indeed, the application of the currently available advanced experimental techniques, that are mainly based on optical methods and have been developed for single phase systems, is not straightforward.

In the past, laser doppler velocimetry (LDV) has been adopted in gas-liquid stirred vessels for the measurement of the liquid flow field by Rousar and Van den Akker (1994) and for the gas flow field by Morud and Hjertager (1996), but it has never been applied to the simultaneous detection of the velocity fields of the two phases that would enable meaningful comparison.

Recently, measurements of the liquid velocity in gassed stirred vessels using a PIV set-up with a single camera have been presented by Aubin *et al.* (2004), Khopkar *et al.* (2003) and Deen and Hjertager (2002). Besides, the development of phase discrimination algorithms in the realm of single camera PIV techniques has been reported in several works (e.g., Delnoij *et al.*, 1999; Linken *et al.*, 1999; Khalitov and Longmire, 2002): though, their application to gas–liquid stirred vessels has been shown to be inadequate, mainly due to the small differences between the liquid and the gas bubble velocities in stirred conditions (Deen *et al.*, 2002a). So far, only Deen *et al.* (2002b) adopted a two-camera PIV method for the simultaneous measurement of liquid and bubble velocities in stirred vessels, but they presented data confined to the impeller blade area. Therefore, sound literature data on the flow field of both liquid and bubbles in turbulent stirred vessels is still quite scant. Indeed, their availability would be beneficial for gaining better insight into the complex two-phase turbulent hydrodynamics, which is a prerequisite for rational design and control of gas-liquid processes, as well as for validating the multiphase models currently adopted in CFD simulations and for their further development.

On the computational side, most of the simulations of gas–liquid stirred vessels have been carried out under simplified assumptions and, due to the lack in experimental data, strict evaluation of prediction accuracy has not been frequently performed.

Deen *et al.* (2002a) have compared radial and axial gas and liquid velocity profiles measured by PIV with the results of Eulerian–Eulerian simulations performed with the CFX-4 code, using the Sliding Grid approach and the $k-\varepsilon$ turbulence model. In the simulations, both bubble diameter and gas-liquid dispersion level were fixed. Discrepancies were found between the PIV and the CFD axial velocity profiles as well as between the observed and calculated gas cavities behind the Rushton turbine blades. A similar modelling approach was adopted by Khopkar *et al.* (2003) for the simulation of a gas–liquid stirred tank equipped with a pitched blade turbine. The evaluation about the suitability of the selected computational approach was mainly based on the comparison with PIV liquid velocity data, that was qualitatively satisfactory, while a quantitative agreement was not obtained for many of the mean and turbulent flow features. Lane *et al.* (2002, 2004) adopted the multi fluid model implemented in the CFX-4 code and coalescence and breakage correlations for the simulation of a gas-liquid stirred vessel, for which bubble distribution profiles were available. With their approach, the prediction of the two-phase flow field, bubble size distribution and gas hold-up was possible, but full agreement with the experiments was difficult to achieve. Moreover, many adjustable parameters were introduced in the models. A more fundamental approach was pursued by Venneker *et al.* (2002), who adopted a strongly simplified two-dimensional description of the flow field and turned their attention to the prediction of the bubble size distribution by means of population balance equations with coalescence and breakage source terms. Other works on the simulation of gas–liquid stirred vessels were performed (e.g., Bakker and Van den Akker, 1994; Morud and Hjertager, 1996), but they were limited to 2-D or based on black-box methods for the impeller description; it is, therefore, difficult to establish whether the rather unsatisfactory results obtained are to be attributed to the two-phase flow modelling or to the failure in the continuous phase flow field prediction, that has a major influence on the gas phase dispersion. In other papers, limited quantitative comparison with experiments has been provided (Ranade and Deshpande, 1999; Khopkar *et al.*, 2006) and only qualitative conclusions can been drawn about the quality of the modelling approaches. Recently, Khopkar and Ranade (2006) and Scargiali *et al.* (2007) have shown that good quantitative prediction of the gas hold-up distribution can be obtained using fully predictive simulations based on the multi fluid model. Overall, the CFD simulations of gas–liquid stirred vessels performed so far have mainly been based on Eulerian models for the two phases and great attention has been devoted to the identification of the most appropriate interaction terms in the momentum equations and, in particular, of the bubble drag law correlation. Nevertheless, more computational work and more extensive and quantitative comparisons with experimental data have to be performed before firm conclusions can be drawn on the most appropriate modelling strategy for gas–liquid stirred vessels.

In this work, the gas–liquid dispersion in a stirred vessel of standard geometry has been studied by a two-phase PIV technique and modelled by RANS-based CFD simulations. The experimental and computational techniques as well as the relevant measured and predicted two-phase flow fields will be discussed in the following.

EXPERIMENTAL AND COMPUTATIONAL METHODS

The investigation was carried out in a fully baffled cylindrical vessel (diameter, $T = 23.6$ cm) provided with a flat base, open on top and filled with demineralised water up to a tank height, H, equal to T. Agitation was provided by a standard Rushton turbine of diameter $D = T/3$ placed at the distance $C = T/2$ from the vessel base. The impeller rotational speed, N, was fixed at 450 rpm, corresponding to the fully turbulent regime and blade tip speed, V_{tip}, equal to 1.85 m s^{-1}. The gas–liquid system consisted of air and water. The gas sparger was made of a tube of 3.3 mm diameter with a porous membrane on top, fixed behind one baffle and feeding the gas below the impeller at a distance equal to $T/4$ from the vessel bottom. The gas flow rate was varied from 0.02 vvm up to 0.5 vvm, corresponding to a gas flow number ranging from 1×10^{-3} to 0.022. These conditions correspond to completely dispersed air bubbles in the liquid phase (Nienow *et al.*, 1985). The vessel was placed in a square cross-section tank filled with the working liquid in order to reduce the laser light refractive effects at the curved vessel surface. The shaft and the impeller were painted matt black to minimise light reflection.

The PIV Technique

The measurements were performed using a Dantec Dynamics PIV system, whose main elements are shown in Figure 1. The laser sheet source adopted was a pulsed Nd:YAG laser, emitting light at 532 nm with a maximum frequency of 15 Hz. The liquid phase velocity was determined from the displacement of fluorescent seeding particles (Rhodamine-B); the relevant image pairs were collected by a Dantec HiSense MK II camera (1344 × 1024 pixels CCD), provided with an orange light filter, that allows the passage of the light scattered by the seeding (wavelength of 600 nm) while blocking lower wavelength light. The gas phase velocity was measured with a PCO Camera (1280 × 1024 pixels CCD), equipped with a green filter, that ensures the transit of the light scattered by the gas bubbles (532 nm) and blockage of that of the seeding particles. The laser control, laser/cameras synchronisation and data acquisition and processing were handled by a hardware module (FlowMap System Hub) and FlowManager software installed on a PC.

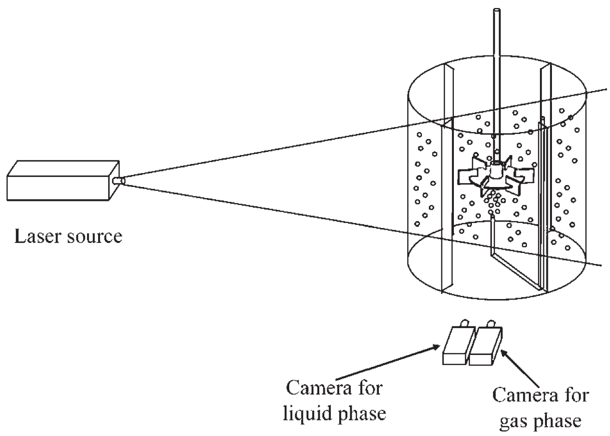

Figure 1. The PIV experimental set-up.