Chinese Journal of Chemical Engineering 23 (2015) 890–896

Contents lists available at ScienceDirect

Chinese Journal of Chemical Engineering

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

Fluid Dynamics and Transport Phenomena

Influence of impeller diameter on overall gas dispersion properties in a sparged multi-impeller stirred tank☆

Yuyun Bao, Bingjie Wang, Mingli Lin, Zhengming Gao, Jie Yang *

State Key Laboratory of Chemical Resource Engineering, School of Chemical Engineering, Beijing University of Chemical Technology, Beijing 100029, China

ARTICLE INFO

Article history:
Received 9 May 2014
Received in revised form 19 August 2014
Accepted 8 November 2014
Available online 21 March 2015

Keywords:
Gas holdup
Mixing
Multiphase reactors
Relative power demand
CFD
Multi-impeller stirred tank

ABSTRACT

The impeller configuration with a six parabolic blade disk turbine below two down-pumping hydrofoil propellers, identified as PDT + 2CBY, was used in this study. The effect of the impeller diameter D, ranging from 0.30T to 0.40T (T as the tank diameter), on gas dispersion in a stirred tank of 0.48 m diameter was investigated by experimental and CFD simulation methods. Power consumption and total gas holdup were measured for the same impeller configuration PDT + 2CBY with four different D/T. Results show that with D/T increases from 0.30 to 0.40, the relative power demand (RPD) in a gas–liquid system decreases slightly. At low superficial gas velocity V_S of 0.0078 $m \cdot s^{-1}$, the gas holdup increases evidently with the increase of D/T. However, at high superficial gas velocity, the system with $D/T = 0.33$ gets a good balance between the gas recirculation and liquid shearing rate, which resulted in the highest gas holdup among four different D/T. CFD simulation based on the two-fluid model along with the Population Balance Model (PBM) was used to investigate the effect of impeller diameter on the gas dispersion. The power consumption and total gas holdup predicted by CFD simulation were in reasonable agreement with the experimental data.

1. Introduction

Gas–liquid dispersion in mechanically agitated vessels is a common operation used in many industrial processes, such as chemical engineering, mineral processing, and wastewater treatment, because it offers unmatched flexibility and control to tailor the fluid dynamics. During the last two decades, more and more researchers have devoted to the study of gas dispersion in stirred tanks [1–3]. In industrial applications, the increase of the reactor scale causes that the height-diameter ratio of vessels could be as large as 2 or 3, which requires multi-impeller configuration for mixing and gas dispersion. Thus, more and more studies were carried out in systems with multiple impellers in recent years [4–9]. Moreover, the optimum design of multi-impeller agitation becomes more and more important for the large scale industrial reactors.

Because of the interactions between fluid and impellers, the flow pattern caused by an impeller can be significantly influenced by another one in a multi-impeller stirred tank. These interactions are greatly affected by the parameters like impeller diameter and spacing [10], leading to different gas dispersion performances in stirred tanks. In large scale industrial gas–liquid reactors, the manufacturing and operational costs are closely related to the optimal design of the impeller, especially the diameter of impellers. Given the same power input into a reactor, the impellers with smaller diameter have higher rotational speed and less torque compared with the impellers of larger diameter. As a result, the sizes of the gearing box, mechanical seals, and impeller shaft are determined after the impeller diameter is specified, which becomes very important in large scale reactors with volume up to 800 m^3. Thus, more research is needed to obtain the relationship between gas holdup and agitator geometrical parameters for the purpose of optimized design of agitators.

During the last two decades, computational fluid dynamics (CFD) techniques have been used to simulate gas–liquid flows in agitated tanks [11–14]. Due to the lack of experimental data, numerical investigation on multiple-impeller systems is less popular than that on single-impeller tanks [15,16]. Moreover, CFD technique can also reveal the detailed information that cannot be easily obtained from experiments, especially the information near the impeller discharging region. Therefore, the application of CFD has obvious advantages in the investigation of gas–liquid flow in stirred tanks.

In the present study, a turbine-propeller combination consisted of a parabolic blade disk turbine below two down-pumping hydrofoil propellers, identified as PDT + 2CBY, was used to investigate the effect of D/T (the ratio of impeller diameter to tank diameter) on power consumption and total gas holdup. Furthermore, the CFD simulation

☆ Supported by the National Natural Science Foundation of China (21121064, 21206002, 21376016).
* Corresponding author.
E-mail address: yangjie1986@mail.buct.edu.cn (J. Yang).

http://dx.doi.org/10.1016/j.cjche.2014.11.030
1004-9541/

was also used to predict the gas dispersion properties in the multi-impeller stirred tank.

2. Experimental

All the experiments were carried out in a stainless steel dished-bottom cylindrical tank with internal diameter $T = 0.48$ m and a filled aspect ratio $H/T = 1.66$, as sketched in Fig. 1. Four baffles each 0.045 m wide were mounted 0.005 m away from the wall. The impeller configuration with a six parabolic blade disk turbine below two down-pumping hydrofoil propeller, identified as PDT + 2CBY (Fig. 2) was used in this study. Impellers with diameters of 0.30*T*, 0.33*T*, 0.37*T*, and 0.40*T* were used. The structural parameters of CBY impeller are shown in Table 1. The distance between neighbor impellers was kept constantly at 0.48*T*. The clearance between the lowest impeller and the base of the tank was 0.33*T*. A ring sparger of 0.8*D* was located 0.25*T* above the tank bottom and with 27 holes whose diameter was 2 mm.

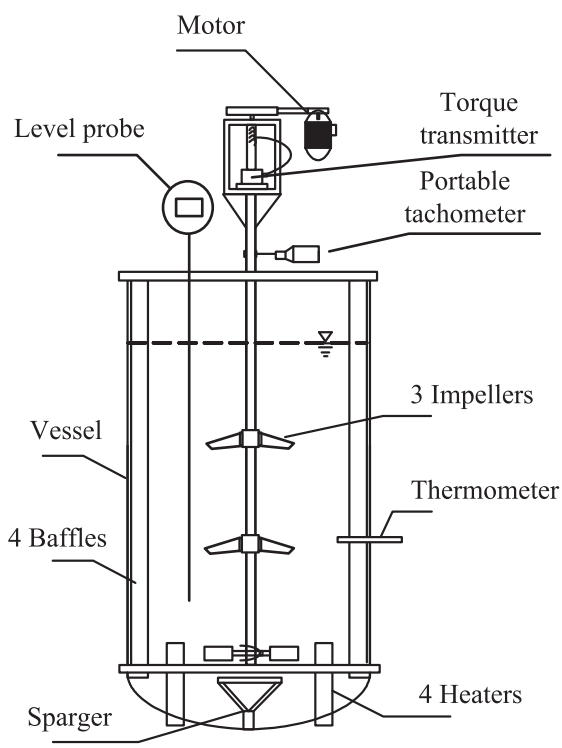

Fig. 1. Schematic of the experimental setup.

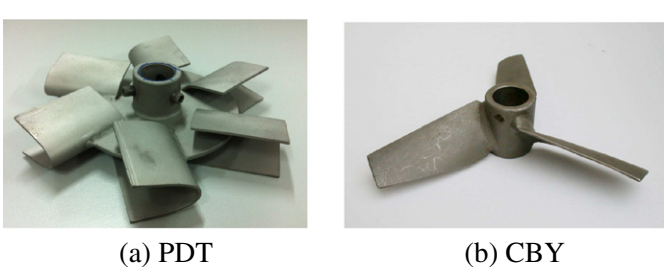

Fig. 2. Impellers.

Air and deionized water were used as the gas and liquid phase in all experiments. Air passed through three stage filters before being sparged into the tank in order to get rid of the impurities in gas. The total gas rates ranged from 5 to 59 $m^3 \cdot h^{-1}$, and the corresponding superficial velocities V_S were from 0.0078 to 0.092 $m \cdot s^{-1}$. The liquid bulk temperature was kept at 24 °C.

Table 1
Structural parameters of CBY impeller

D/T	Diameter/mm	Blade		$Angle_{tip}$/(°)
		Length/mm	$Width_{tip}$/mm	
0.30	141	59.5	14.8	22.6
0.33	155	66.0	16.8	23.0
0.37	174	75.7	18.5	21.9
0.40	189	81.9	20.1	22.7

The power consumption was calculated from the torque and rotational speed of the shaft measured with a torque transmitter and a portable tachometer, respectively. The gas holdup was calculated from the changes in liquid level measured by a calibrated radar probe (Krohne Reflex-Radar BM100A, Germany). The total gas holdup is defined as

$$\varepsilon = \frac{H_g - H_0}{H_g - H'} \tag{1}$$

where H_g, H_0 and H' are respectively the liquid level of air-sparged tank, static liquid level and elliptic tank bottom correction coefficient which is equal to 0.04 m.

3. Computational Model

In an aerated stirred tank, the Eulerian–Eulerian approach is a reasonable choice for its ability to handle the system with large and low volume fractions with acceptable accuracy. The standard k-ε model was applied with its standard constants to simulate the continuous liquid-phase turbulence. For the dispersed gas phase, the zero equation model was used with the Sato enhanced eddy viscosity model [17] which included the influence of large bubbles on the liquid-phase turbulence.

The total interfacial force between the two phases may arise from several independent physical effects, such as the interphase drag, lift, wall lubrication, virtual mass forces. According to the analysis of Khopkar *et al.* [18] some forces such as lift force and virtual mass force can be neglected in the case of stirred vessel. So we only considered the drag and turbulence dispersion forces. The modified Grace model [19], which used a simple power law correction to amend the single bubble Grace drag coefficient for high bubble volume fractions, was used to calculate the interphase drag force. The model of Lopez de Bertodano [20] was used to compute the turbulent dispersion force.

According to the analysis of Min *et al.* [16] the single average bubble diameter (SABD) approach could not correctly predict the gas void fraction whether for the high-shear-rate region near the lowest impeller or the low-shear-rate region near the free surface. In order to predict the performance of gas–liquid stirred tank accurately, the distribution of bubble size should be considered. The population balance (PB) equations could be used to simulate the non-uniform bubble size distribution in a stirred vessel. In this paper, multiple size group (MUSIG) Model with PB model in CFX were used to handle poly-dispersed multiphase flows. The MUSIG model was a framework in which PBs were solved simultaneously with the Navier–Stokes equations for Eulerian gas and liquid phases. Population balances including breakup and coalescence effects provided a well-established method to calculate the changing size distribution of a poly-dispersed phase. The MUSIG model assumed that all bubble size groups shared a common velocity field but with different slip velocities for different size bubbles. The drag force on bubbles was calculated based on the local Sauter mean bubble diameter.