Journal of the Taiwan Institute of Chemical Engineers 000 (2017) 1–11

Contents lists available at ScienceDirect

Journal of the Taiwan Institute of Chemical Engineers

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

A CFD study of the drawdown speed of floating solids in a stirred vessel

Ruey-Chi Hsu*, Chun-Kai Chiu, Shang-Chain Lin

Department of Chemical and Materials Engineering, Chang Gung University, 259 Wen-Hwa 1st Road, Kwei-Shan, Taoyuan 33302, Taiwan, ROC

ARTICLE INFO

Article history:
Received 8 July 2017
Revised 14 December 2017
Accepted 15 December 2017
Available online xxx

Keywords:
Floating solids drawdown speed
Stirred vessel
CFD
Baffle
Impeller

ABSTRACT

The drawdown of floating solids is studied through visual observation and CFD simulations in this work. Four different baffle configurations and two different impellers were used to study the effects of the baffle design, the clearance size and the impeller type on the just drawdown speed of floating solids in a stirred vessel. Particles accumulate in the vicinity of the shaft above the impeller level in the Rushton impeller system at higher speeds. A full top-to-bottom circulation loop was found in the A310 impeller system at higher speeds. Increasing the clearance can lead to a more homogeneous particle concentration distribution in the stirred vessel. The shift baffles and hollow-out baffles show superior particle drawdown performances than the standard baffles. It was also found that higher clearances result to a better drawdown rate of floating solids for both impellers. A new method has been proposed to determine the just drawdown speed based on the rotating impeller domain in the CFD simulations. The CFD predicted just drawdown speeds agree well with our experimental observations.

1. Introduction

The drawdown of floating solids in a stirred vessel is often required in numerous industries, including fermentation, pharmaceutical, food, paint, polymerization, and sewage treatment industries. In such an operation, the design objective is to achieve maximum wetting between the solid and liquid. When focusing on the evaluation of the performance of the drawdown of floating particles, the type of impeller, the impeller clearance, the baffle configuration and the particle floating characteristics are all key variables to consider.

Edwards and Ellis [1] first investigated the critical suspension of the floating particle in a solid–liquid two-phase system. They found that various types of impellers have a strong influence on the suspension of the floating particles. When testing three types of impellers (turbine, propeller, and paddle), the paddle required the lowest speed, while the propeller required the highest speed to reach the critical suspension. In a gas–liquid-floating particle three-phase system multi-impellers system, the critical suspension speeds of floating particles were measured by Xu et al. [2]. They investigated four types of impellers: simple axial-flow impeller upflow (SPU) and downflow (SPD), disk turbine (DT) and wing turbine (WT). Bakker and Frijlink [3] also showed that the mixed flow impeller, Pitch Blade Turbine (PBT), was more energy-efficient in achieving complete particle drawdown than radial flow impellers. There is a considerable amount of literature done on PBT type impellers. Ozcan-Taskin and Wei showed that the floating particles drawdown paths of up-pumping hydrofoils were dependent on the number of baffles, the impeller pumping mode, and submergence levels [4].

The designs of the baffles have been used to increase the flow in an agitated vessel and improve the drawdown of floating particles. Previous investigations showed that a narrow set of four baffles, *i.e.*, approximately 2% to 2.5% of the vessel's diameter, was recommended in particular for the drawdown of particles. A narrow baffle increased mechanical stability, lowered drawdown power requirements, provided uniform solids distribution and showed a limited surface vortex in the vessel [5]. Furthermore, the drawdown power requirements of the narrow baffle system was relatively unaffected by impeller submergence [5]. The down-triangular baffles were probably the best configuration as they performed better than both the narrow baffles and standard baffles in terms of floating particle drawdown and allowed for more flexibility in impeller placement in the vessel to avoid air entrainment [6]. The mechanisms and the hydrodynamics of particle drawdown in numerous design systems are very complex. As a result, Computational Fluid Dynamics (CFD) is often used as a powerful tool to study particle drawdown in stirred tanks since 1980s [6–16]. Numerous models and solution techniques have been developed over the years and are well documented in a wide variety of mixing problems. Many of these CFD results have been verified by experimental results.

* Corresponding author.
E-mail address: rchsu@mail.cgu.edu.tw (R.-C. Hsu).

https://doi.org/10.1016/j.jtice.2017.12.014
1876-1070/© 2017 Published by Elsevier B.V. on behalf of Taiwan Institute of Chemical Engineers.

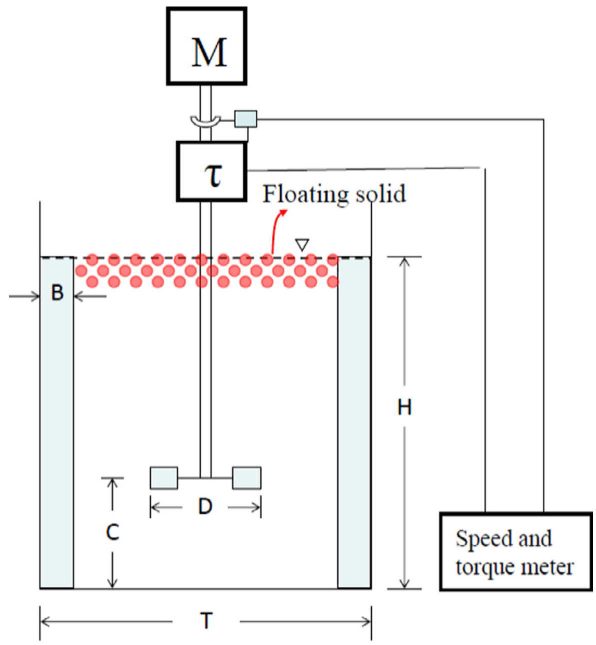

Fig. 1. Schematic diagram of the experimental setup.

The objectives of this paper are to investigate the effect of baffle configurations, and impeller clearance on the drawdown of floating particles in a stirred vessel with two different impellers, Rushton and A310, through visual observation and CFD simulations. Four baffle configurations are proposed: standard baffles, upper cut baffles, shift baffles, and hollow-out baffles. The just drawdown impeller speed in terms of the particles density of rotating frame is also discussed.

2. Experimental

The schematic of the experimental setup where the experiments were conducted is shown in Fig. 1. The mixing the solid/liquid system was carried out in a cylindrical flat-bottomed mixing vessel (tank diameter, $T = 0.30$ m, liquid level, $H = T$) made of transparent acrylic with four baffles of standard size, B, as $T/10$. The suspension solution was stirred by centered impeller and the clearance, C, was $= T/3$ or $T/2$. Two different impellers were used in this study, including the radial flow 6-blade standard Rushton turbine, and the axial flow A310 impeller (Ligtnin). The working diameter of the impeller, D, was set as $T/3$. The impeller was driven by a inverter duty motor. The impeller torque was detected by a detector (SS-020, ONO SOKKI) followed by a torque meter (TS-200, ONO SOKKI). Four baffle configurations were used in this study, including the standard configuration, the upper-cut configuration, the shift configuration and the hollow-out configuration. Their geometries and their positions relative to the tank wall (*i.e.*, the red line in Fig. 2) are shown in Fig. 2.

Mono-sized 355 µm polyethylene particles were used as the testing floating particles in water. They were dyed red to ease the visual observations. The particle density was 469 kg/m^3. The bulk solid concentration in water was 1 wt%. The particle just drawdown state was defined as the state of the system at which no particles stayed at the surface longer than 2 s. The just drawdown speed, N_{jd}, was defined as the impeller speed at the just drawdown state. It was difficult to determine because even at a higher impeller speed some solid particles reappeared on the liquid surface. Experimental N_{jd} was determined by averaging N_{jd} from three independent runs. The error was estimated as ±5 rpm.

3. CFD simulation

The CFD simulation studies used the commercial software package, Ansys Fluent Version 16.2. The system of interest was generated by the Ansys Design Modeler software and the grid generation used the multi-block method (Ansys Meshing). The numbers of the CFD simulation grids for different configurations were listed in Table 1. The number of the grids was varied from *ca.* 498,000 to 582,000. Two domains were set in the system: the central rotating impeller domain and the stationary baffle domain. The central impeller domain was in a rotating frame of reference and the stationary baffle domain was in a stationary frame of reference. Fig. 3 shows two representative cases to show the two domains of interests. The rotating impeller domain is shown in Fig. 3(a) and (b) and the coupling of the two domains is shown in Fig. 3(c) and (d) using the standard baffle configuration as the example. The relative motion between the rotating impeller domain and the stationary baffle domain was realized by the MRF approach [17]. The initial condition for the simulation was that of still liquid and solid particles floating at the top of the computational domain. The free liquid surface was treated as the symmetry boundary condition. All the cases were modeled as steady state, and they were assumed

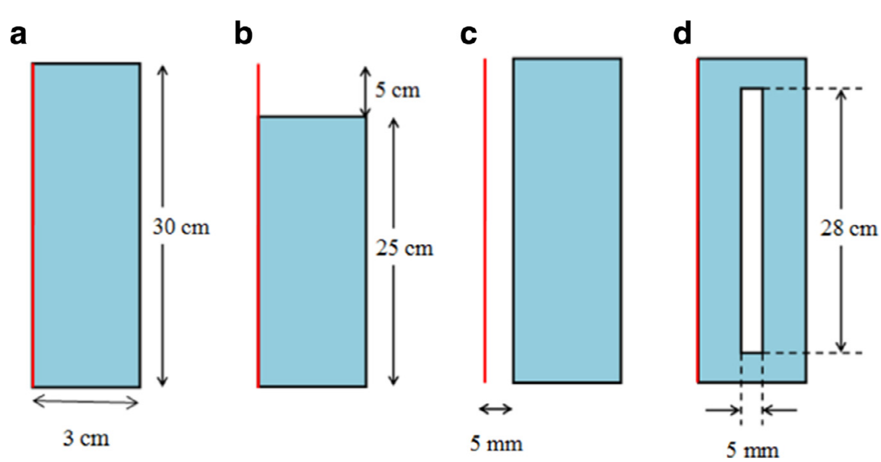

Fig. 2. The configuration of baffles: (a) standard, (b) upper-cut, (c) shift and (d) hollow-out. (For interpretation of the references to color in this figure, the reader is referred to the web version of this article.)