Fluid dynamics and mixing of single-phase flow in a stirred vessel with a grid disc impeller: Experimental and numerical investigations

V. Buwa*, A. Dewan1, A.F. Nassar, F. Durst

Lehrstuhl für Strömungsmechanik (LSTM), Friedrich-Alexander-Universität Erlangen-Nürnberg, Cauerstraße 4, D-91058 Erlangen, Germany

Received 21 April 2005; received in revised form 4 October 2005; accepted 6 October 2005

Available online 4 January 2006

Abstract

We report experimental and numerical investigations of a novel grid disc impeller for mixing of single-phase flow in stirred vessels. We performed detailed mean velocity measurements using LDA to understand the flow generated by the grid disc impeller and we also measured the mixing time and power consumption of the grid disc impeller. Measurements showed that the performance of the grid disc impeller, which has radial flow characteristics, is equivalent to a standard propeller in terms of the degree of mixing achieved per unit power consumption. We also performed numerical simulations to predict the flow generated by the grid disc impeller and its mixing performance. It was shown that the present computational model predicts the mean velocities, power consumption and mixing time in good agreement with the measurements. The experimentally validated computational model was further used to understand the effects of impeller rotational speed and grid disc configuration on the fluid dynamics and mixing performance of the grid disc impeller.

© 2005 Elsevier Ltd. All rights reserved.

Keywords: Grid disc impeller; Fluid mechanics; Mixing; Simulation; Single-phase flow; Stirred vessel

1. Introduction

Stirred vessels are widely used in the chemical, mineral processing, wastewater treatment and several other industries. A large number of process applications involve mixing of single-phase flow in mechanically stirred vessels. The energy requirements of different mixing processes constitute a major expenditure and therefore more profitable operation protocols need to be developed. Several efforts have been made in the past to improve the performance of different impellers in terms of their power consumption, pumping capacity and degree of mixing achieved and new types of impellers are continuously being evolved (see, for example, Li et al., 2004; Mavros et al., 2001). However, it appears that all the energy provided by the impeller to the fluid is not efficiently utilized to the desired mixing in stirred vessel.

It is generally believed that mixing in a stirred vessel is dominated by the turbulent diffusion (or dispersion) compared to the convective mixing. However, some recent investigations in the literature (for example, Patwardhan et al., 2003) indicated that the convective mixing is the dominant mechanism of mixing in stirred vessel. Therefore, an energy efficient mixing impeller can be defined as the one that can convect and disperse the tracer optimally at low power inputs. In the present work, therefore, an attempt has been made to use a grid, which is known to modify the local turbulence due to its shape, as an impeller. It will be shown in the following sections that such grid disc impeller can produce local turbulence and convective flow at low power inputs to provide energy efficient mixing in stirred vessels.

Mixing in a stirred vessel is achieved by three means: at the molecular level, due to the turbulent fluctuations and due to the bulk convection or circulation of the fluid. Various parameters, such as the impeller shape, impeller diameter, number of impeller blades, tank diameter, clearance of impeller from the tank bottom are known to affect significantly the flow within the vessel and hence the mixing characteristics. Depending on the
process needs, several types of impellers are used in industry. These impellers are broadly classified as radial (for example, Rushton turbine) or axial (for example, propeller and pitched blade turbine), depending on the direction of the secondary flow (the primary flow is the rotational flow in the direction of impeller motion). The direction of the secondary flow significantly influences the mixing in stirred vessels. These standard impellers have been widely investigated (for example, see Lee and Yianneskis, 1998; Schafer et al., 1998; Derksen et al., 1999; Aubin et al., 2001).

The fluid dynamics of flow in stirred vessels is extremely complex and therefore is not well understood. The flow around a rotating impeller interacts with baffles in the tank, resulting in unsteady, three dimensional, rotational turbulent flow. With continuous research efforts over the last decade, computational flow modeling has now become an effective tool for the design of stirred vessels. One of the challenges to the numerical simulation of the flow in a stirred vessel is the simulation of the fluid turbulence. One of the ways to simulate turbulent flows is to perform large eddy simulation (LES) or direct numerical simulation (DNS). However, simulations of a fully turbulent flow with very high Reynolds number (typically 50,000 and higher) as encountered in practical applications require computational resources that are prohibitively expensive. Therefore, DNS and LES cannot be used as a tool for the design of impellers (Montante et al., 2001).

The other approach is to use the Reynolds-averaged Navier–Stokes (RANS) equations, and this is the most widely used approach (see, for example, Khopkar et al., 2004; Li et al., 2004). Several types of turbulence models can be used to close the RANS equations, which include the simplest zero-equation model to the most complex Reynolds-stress transport models. The standard $k–\epsilon$ model of turbulence has been widely applied to study such flows (Khopkar et al., 2004). Jenne and Reuss (1999) and Mueller and Schaefer (2003) have shown that the modification of the standard $k–\epsilon$ model to account for the effect of anisotropy does not improve the predictions. Further, it is well known that the use of much more complex turbulence models, such as low Reynolds number versions of the $k–\epsilon$ model (Jones et al., 2001) and Reynolds-stress transport models, does not significantly improve the predictions of turbulent flows compared with the standard $k–\epsilon$ turbulence model (see, for example, Montante et al., 2001). It may be noted that several types of Reynolds-stress transport models and low Reynolds number versions of the $k–\epsilon$ model exist in the literature and it is not clear which one is the most suitable for complex flows, such as that encountered in stirred vessels.

1.1. Present work

We report experimental and numerical investigations of the flow generated by a new grid disc impeller designed to achieve efficient mixing in stirred vessels. The profiles of mean velocity components and turbulent kinetic energy at different locations in the vessel were measured using laser Doppler anemometry (LDA). Further, we measured the mixing time and power consumption to evaluate the performance of the grid disc impeller against the conventionally used impellers. Numerical simulations of the flow generated by the grid disc impeller were performed using a commercial flow solver CFX 5.7 (ANSYS Inc., USA) and predictions were validated using measured profiles of mean velocity and turbulent kinetic energy. The experimentally validated computational model was further used to study the effects of rotational speed and grid disc configuration on the power consumption and mixing achieved by the grid disc impeller.

2. Experimental

2.1. Experimental set-up

Measurements were performed in two cylindrical vessels with different configurations. For the LDA measurements, a small baffled vessel [diameter ($T$) = 150 mm and liquid height $(H)$ to $T$ ratio of 1] with four baffles (baffle width 15 mm, thickness 2 mm and wall clearance 2.6 mm) as shown schematically in Fig. 1 was used. A radial grid disc impeller [diameter $(D) = T/3$], as shown in Fig. 1 (inset), with a bottom clearance of $T/3$ was employed. The measurements were performed using “white paraffin” oil ($\rho = 839 \text{ kg/m}^3$ and $\mu = 12.589 \times 10^{-3} \text{ kg/m.s}$) at an impeller rotational speed of 1500 rpm (corresponding to a tip velocity of 3.93 m/s) and an impeller Reynolds number $(Re = ND^2/v)$, where $N$ is the impeller rotational speed and $v$ the kinematic viscosity of the liquid) of 4084. Measurements of the radial variations of the axial, radial and tangential components of mean velocity and turbulent kinetic energy at different axial positions in the vessel were carried out. The purpose of these experiments was to understand the flow generated by the grid disc impeller and to generate experimental data for the validation of the numerical simulations carried out in this work. The use of a smaller vessel in the validation study permitted investigations of the effect of different grid resolutions on the computed flow field with fine grid resolutions that would have been difficult to employ for a large vessel.

The mixing performance and power consumption of the grid disc impeller were studied in a larger vessel ($T = 300 \text{ mm and } H/T = 1$) with four baffles (baffle width 23 mm, thickness 2 mm and wall clearance 4 mm). Experiments were carried out using two types of grid disc impellers, a radial grid disc and square grid disc, as shown in Figs. 2(a) and (b), respectively. A bottom impeller clearance of $T/3$ was used in all experiments. In the experiments carried out to measure power consumption and mixing time, tap water under ambient conditions was used as the working fluid. Experiments were also performed using a solid disc $(D = T/3$ and thickness $= 1 \text{ mm})$ and a standard propeller $(D = T/3$ [Figs. 2(c) and (d)]) to evaluate the performance of the grid disc impeller. The effects of the impeller rotational speed (in the range 180–360 rpm) and grid disc configurations on the power consumption and mixing time were also studied. The details of different measurement techniques used are discussed in the following sections.
2.2. LDA measurements

A multifunctional stirrer-LDA test rig (Fig. 1) was used to measure mean velocity profiles. The LDA head was supported on a three dimensional traversing system, which was driven by three step motors and controlled by a computer (with an accuracy of 0.1 mm in three directions). The LDA system employed was a diode fiber laser Doppler anemometer (DFLDA). It works in the backward scattering mode and has a control volume diameter of 60 μm and a length of 1.53 μm. The disadvantage due to the low scatter intensity in the backward scattering mode was compensated by seeding into the working fluid silver-coated hollow glass spheres (DANTEC DYNAMICS GmbH). The spheres had a density of 1400 kg/m³ and an average diameter of 10 μm, ensuring satisfactory agreement between particles and corresponding flow velocities. A measuring rate in the range 30–150 Hz was obtained depending on the particle concentration and the measuring positions in the vessel. The complete measuring section was refractive index matched in order to eliminate distortion effects of the path of the laser beams and the scattered light. For this purpose, the cylindrical vessel, the baffles and the stirrer blades were constructed from Duran glass that had the same refractive index (n = 1.47) as the working fluid. In addition, the stirred tank was located in an outer container filled with the working fluid. To avoid surface aeration by the applied high velocity, the vessel was covered from the top. Further details of the LDA measurements can be found in Nassar (2005).

2.3. Power consumption and mixing-time measurements

The power consumed by the impeller was measured using a torque meter (HBM, Germany). Mixing time measurements were carried out by instantaneous addition of a tracer and subsequently monitoring the time of evolution of the tracer concentration at a certain location in the vessel using a conductivity probe (WTW, Germany). A 0.1N salt solution (NaCl) was used as a tracer and 30 ml (corresponding to 0.136% of the tank volume) of the tracer solution was injected inside the vessel near the shaft using a syringe. The mixing time (θ) was calculated as the time required for the tracer concentration to reach within 100 ± 5% of the final (equilibrium) tracer concentration. The mixing time and power consumption experiments were measured under different impeller rotational speeds for different impellers described in Section 2.1 and results of these measurements are discussed in Section 4.

3. Numerical simulations

In the past, three different approaches were used to model the rotational motion of an impeller: (1) black box approach, (2) multiple frames of reference approach and (3) sliding mesh approach. Detailed information on these approaches with
In the literature (Brucato et al., 1998; Ranade, 2002). In the reference to their advantages and disadvantages can be found in the literature (Brucato et al., 1998; Ranade, 2002). In the present work, we used the ‘multiple reference frames’ (MRF) approach to simulate the motion of the rotating impeller. In the MRF approach (Luo et al., 1994), the solution domain is divided into two regions, one containing the impeller (rotor region) and the other containing the tank and baffles (stator region).

In this work, numerical simulations of the flow generated by the grid disc impeller were carried out by solving the RANS equations. Different sets of governing equations were solved in the tank and impeller regions using the stationary and rotating frames of reference, respectively (see Brucato et al., 1998; Ranade, 2002 for details of the governing equations). In the rotating frame of reference attached to the rotor, the momentum equations contain additional terms due to the centrifugal and Coriolis forces. The standard k-ε model was used to simulate the fluid turbulence with the model constants $c_{1k} = 1.44$, $c_{2k} = 1.92$, $c_{μ} = 0.09$, $σ_k = 1.0$ and $σ_ε = 1.3$ (Lauder and Spalding, 1974). The momentum conservation equation was closed by calculating the effective viscosity $μ_{eff}$ from the turbulent kinetic energy and its dissipation rate as $μ_{eff} = μ + C_μk^2/ε$ where $μ$ denotes the molecular viscosity.

The governing equations were solved using the commercial flow solver CFX 5.7 (ANSYS Inc.). Owing to the symmetrical features of the stirred vessel equipped with four baffles and a grid disc impeller about the axis of rotation, one-quarter of the stirred vessel was considered for the numerical simulations with the rotationally periodic boundary conditions at the corresponding faces in the impeller and tank regions [see Fig. 3(a)]. No slip boundary conditions were imposed at the solid walls of the tank and the grid disc. The scalable wall functions were employed to account for the effects of the solid walls (CFX 5.7 User Manual).

A hybrid unstructured mesh (composed of hexahedral elements in the stator region and tetrahedral elements in the rotor region) was generated using commercial grid generation software ICEMCFD-5.0 (ANSYS Inc.). In order to resolve the flow through the impeller grid openings accurately, a sufficiently large number of cells were used in the rotating impeller region. In the present simulations, this region extended from 37.5 < z < 62.5 mm and 0 < r < 37.5 mm for the small vessel with $T = 150$ mm (and 75 < z < 125 mm and 0 < r < 75 mm for the large vessel with $T = 300$ mm). Simulations were performed with different grid resolutions to achieve grid-independent predictions. These results are discussed in Section 4.2.

Depending on the gradients of the mean flow variables, a combination of the first-order and second-order schemes was used to discretize the spatial derivative terms in the governing equations (CFX 5.7 User Manual and reference cited therein). For the discretization of the temporal derivatives, a second-order scheme was used. In all simulations, computations were performed until the residuals of mean velocities, pressure and turbulence quantities were less than $10^{-6}$. An automatic time scale was used for obtaining the time step, which estimates the step based on the flow characteristics. All computations were performed at the Regional Computing Centre (RRZE) of the University of Erlangen–Nürnberg. A typical simulation performed using four Itanium processors in parallel took approximately 144 h of CPU time to achieve the converged solution.

4. Results and discussion

4.1. Flow generated by the grid disc impeller

First, we carried out numerical simulations to study in detail the flow generated by a radial grid disc impeller in a small vessel ($T = 150$ mm; see other details given in Section 2.1). Typical mean flow field obtained by using the radial grid disc impeller at a rotational speed of 1500 rpm at an r-z plane [22.5° behind the baffle in the direction of impeller rotation (anticlockwise)] is shown in Fig. 3(b)(i). The two circulation regions,
which correspond to secondary flow pattern superimposed on primary rotating flow, can be clearly seen from the vector plot in Fig. 3(b)(i). Fig. 3(b)(ii) shows that the magnitude of the mean flow through the impeller openings increases with the radial distance from the axis of the shaft.

4.2. Effect of grid resolution and experimental validation

After performing the preliminary numerical simulations to understand the flow generated by the grid disc impeller, we performed numerical simulations using different grid resolutions obtained by using 22,498 cells (coarse), 46,953 cells (medium), 93,440 cells (fine) and 175,661 cells (very fine) to study the sensitivity of the predicted results to the grid resolution. These simulations were performed for the small vessel with \( T = 150 \) mm and at an impeller rotational speed of 1500 rpm for which the measurements of profiles of mean velocity components and turbulent kinetic energy were made.

The effect of the grid resolutions on the predicted mean axial velocity profiles (normalized with the impeller tip velocity) was studied at three different axial locations that correspond to lines L2, L4 and L6 [at axial positions of 25, 52 and 100 mm, respectively, and all lines are at \( \theta = 90^\circ \) as shown in Fig. 3(a)]. A comparison of the predicted and measured axial velocity profiles at line L4 (just above the impeller) shows that the predicted profiles are not affected by the grid resolution [Fig. 4(a)]. The predicted axial velocity profiles agree satisfactorily at the axial location just above the impeller [Fig. 4(a)], except in the close vicinity of the impeller. The liquid jets emanating from the openings of the radial grid can be clearly seen from the predicted axial velocity profiles [Fig. 4(a)]. The predicted normalized radial velocity profiles for the line L4 are sensitive to the grid resolution [see the predictions for the coarse and medium grids in Fig. 4(b)]. However, with further grid refinement, the predictions become independent of grid resolution. The trends of the tangential velocity profiles [Fig. 4(c)] are similar to those for the radial profiles [Fig. 4(b)]. The agreement between the predicted and measured axial, radial and tangential profiles at two other axial locations (at lines L2 and L6) was found to be satisfactory, except in the close vicinity of the tank wall (not shown here).

Although the trends of variations in the turbulent kinetic energy and the locations of its maximum values in the tank were predicted qualitatively well, the magnitudes of the turbulent kinetic energy were under-predicted [Fig. 4(d)]. It may be noted that several previous investigators have reported similar
We measured power consumption and mixing time of different impellers (shown in Fig. 2) for the impeller rotational speeds in the range 180–360 rpm (corresponding impeller Reynolds numbers in the range 30,000–60,000) and the results are shown in Fig. 5. We also performed measurements with a standard propeller, which is known to have low-power requirements, and also with a solid disc impeller as a limiting case with the same diameter as that of the grid disc impeller (Fig. 5). For all impellers considered in the present work, the mixing time decreases as the impeller speed (or the power input) is increased and this behavior conforms well with reports in the literature (see, for example, Nere et al., 2003 and references cited therein). Further, it can be seen that the mixing performance of the radial grid disc is as good as that of the standard propeller for the equivalent power consumption. On the other hand, the solid disc has a much larger mixing time for the corresponding power inputs. It can be seen that changes in the grid disc configuration do not significantly influence the mixing performance of the grid disc impeller.

We performed numerical simulations to understand further the effects of impeller rotational speed and grid disc configuration. The mixing simulation was started by the instantaneous addition of a tracer near the shaft slightly below the water level in the tank after a steady flow solution had been obtained. During the mixing simulation, only the unsteady species transport equation was solved using the already converged mean flow fields. It may be noted that the mixing time values based on the tracer concentration-time history recorded at any particular location in the vessel can be sensitive to the location of the probe and therefore may not provide the correct overall mixing time. Therefore, to avoid such a discrepancy, eight monitor points distributed over the entire solution domain were considered for the calculation of the mixing time. Typical evolution of tracer concentrations at these locations is shown in Fig. 6 for the radial grid disc impeller rotating at a speed of 240 rpm. The mixing time was calculated from the tracer time history recorded at each monitoring point as the time for the tracer concentration to lie within 100 ± 5% of the final (equilibrium) tracer concentration. The average of the mixing times calculated at different monitor points and the spread in the mixing time about this average value were used to characterize the mixing process. The effects of the impeller rotational speed and grid disc configuration on the power consumption and mixing time were studied and the results are presented in the following sections.
The effect of the impeller speed on the flow behavior in the stirred vessel for impeller rotational speeds of 180, 240, 300 and 360 rpm was studied. With increase in the impeller rotational speed, the axial velocity profiles normalized the tip velocity remained unaffected and indicated that the flow characteristics remain almost unchanged with increase in the rotational speed. Similar trends were observed for the predicted radial and tangential velocity components and turbulent kinetic energy. The predicted power number (defined as \( N_P = P / \rho N^3 D^5 \), where \( P \) is the power consumed by the impeller and \( \rho \) is the density of liquid) are in good agreement with measurements for two configurations of the grid disc impeller at all rotational speeds considered (Fig. 7). It can be seen that the measured dimensionless mixing times (\( N\theta \)) agree well quantitatively in the range of mixing time distributions predicted for different rotational speeds of the radial grid disc and square grid disc (Fig. 8). As seen from Fig. 6, the mixing time depends on the tracer monitoring location and therefore the mixing time values shown in Fig. 8 represent an average of those obtained at eight monitoring locations in the vessel. The spread of the mixing time about the mean value is indicated by the error bars as shown in Fig. 8.

### 4.3.2. Effect of grid disc configuration

We also performed the numerical simulations to investigate the effect of grid disc configuration on the fluid flow, power consumption and mixing performance of the grid disc impeller. Two different configurations of the grid disc impeller, a square grid disc with a square opening of 5 mm pitch and a radial grid disc, were considered (Fig. 2). From the comparison of the axial velocity profiles at different axial locations obtained for these two disc configurations (not shown here), it was observed that the flow behavior in the bulk of the tank is almost independent of the configuration of the impeller. However, the flow near the impeller showed a dependence on the impeller configuration. The predicted power consumption and mixing time also agree well with the experimental measurements for the two grid disc configurations considered (Figs. 7 and 8). Although no significant difference was observed in the flow generated and mixing achieved by the radial and square grid impellers, further simulations indicated that the grid opening size and the impeller diameter significantly improve the performance of the radial grid disc impeller. Further numerical investigations are in progress to optimize the performance of grid impeller and will be reported separately.

### 5. Conclusions

A new grid disc impeller has been proposed. The flow characteristics of this impeller under a wide range of operating conditions were studied in detail both numerically and experimentally. The three dimensional rotational turbulent flow was computed using the Reynolds-averaged Navier–Stokes equations. The standard \( k-\varepsilon \) model of turbulence along with wall functions was used to account for the effect of turbulence in the flow. The multiple frames of reference approach was used to model the impeller rotation with respect to the stationary vessel and its components. Measurements of profiles of mean velocity components, turbulent kinetic energy, mixing time and power consumption were performed and compared with the predictions. An overall good agreement between the predictions and measurements was observed. The present study shows promising results for the new impeller as far as the mixing time and power consumption are concerned. The performance of the proposed grid disc impeller is comparable to that of a standard impeller which is known to have low power requirement.

### Acknowledgments

V.B. and A.D. gratefully acknowledge the Bavarian Science Foundation and Alexander von Humboldt Foundation, respectively, for the research fellowships. We thank INVENT Umwelt...
References


