Data Article

Data on the hydrodynamics and power consumption induced by modified anchor impellers in cylindrical tanks

Yousef Kamla\textsuperscript{a,*}, Houari Ameur\textsuperscript{b}, Mohammed Ilies Arab\textsuperscript{c}, Belalia Azeddine\textsuperscript{a}

\textsuperscript{a} Faculty of Technology, University Hassiba Ben Bouali of Chlef, Algeria  
\textsuperscript{b} Department of Technology, University Centre of Naama, P.O. Box 66, Naama 45000, Algeria  
\textsuperscript{c} Laboratory of Marine Science and Engineering, Faculty of Mechanical Engineering, USTO-MB, Oran, Algeria

\section{Article Info}

\begin{itemize}
  \item Article history:
  \begin{itemize}
    \item Received 8 January 2021
    \item Revised 25 November 2021
    \item Accepted 30 November 2021
    \item Available online 3 December 2021
  \end{itemize}
  \item Keywords:
  \begin{itemize}
    \item Stirred tank
    \item Anchor impeller
    \item Newly designed impellers
    \item Fluid flows
    \item Power consumption
  \end{itemize}
\end{itemize}

\section{Abstract}

In the present paper, the data assembled concerning the stirring of a viscous Newtonian fluid in cylindrical and unbaffled tanks is disclosed. The stirring is ensured by close-clearance impellers rotating at low speeds. This is a comparative study between three modified geometries of anchor impellers aiming to enhance the fluid circulation in the whole vessel volume, and especially at the lower part of the vessel. The suggested geometrical configurations aim also to keep the energy consumption at its minimum. The data summarized here provides an additional knowledge for the best selection of stirrers for a specified industrial application.

© 2021 The Author(s). Published by Elsevier Inc. This is an open access article under the CC BY license (http://creativecommons.org/licenses/by/4.0/)

* Corresponding author.  
E-mail address: y.kamla@univhb.dz (Y. Kamla).

https://doi.org/10.1016/j.dib.2021.107669
2352-3409/© 2021 The Author(s). Published by Elsevier Inc. This is an open access article under the CC BY license (http://creativecommons.org/licenses/by/4.0/)
Specifications Table

| Subject                      | Chemical engineering                  |
|------------------------------|---------------------------------------|
| Specific subject area        | Fluid dynamics                        |
| Type of data                 | Tables, Figures, text file.           |
| How data were acquired       | 3D numerical simulation of the flow in a stirred tank using the software (ANSYS CFX) |
| Data format                  | Analyzed                              |
| Parameters for data collection | CFD package used–Ansys CFXMethod–Finite volume approachComputer used for simulation–Processor (I7, 2.5 Ghz).Fluid parameters: |
|                             | • Density: 1.26 kg/m³                 |
|                             | • Viscosity: 1.49 kg/m s              |
|                             | • Laminar regime: Re = 10-100         |
| Description of data collection | Data were collected by means of numerical simulations, the RRF (Rotating reference frame) method was used. This last is very common in CFD applied to agitated tanks, it gives very accurate results and consists in fixing the impeller and considering the fluid domain as rotating around the impeller (considered as a relative reference frame). |
| Data source location         | Faculty of Technology, University Hassiba Ben Bouali of Chief, Algeria. |
| Data accessibility           | Data is given in this paper. In addition, the CSV files of Fig. 5 are |

Value of the Data

• The data reveal the efficiency of newly designed anchor impellers in agitated tanks.
• The reported research is valuable for industrial processes involving homogenization of highly viscous liquids in mixing systems.
• The data provide information on the hydrodynamic and power consumption within vessels agitated by various designed impellers.

1. Data Description

The agitation in vessels is a widespread operation across different industries, technically easy to realize but difficult to characterize [1,2].

The data obtained regarding the agitation of viscous Newtonian fluids in cylindrical tanks is analyzed in this paper. A comparison is performed here between various geometrical configurations of anchor impellers. This geometrical design is generally employed for the agitation of highly viscous fluids at low Reynolds numbers (i.e., at low rotational speed), the so-called laminar flow regime [3–5].

Three Tables and seven Figures are included and which summarize useful information on the flow patterns and power input for such devices. Table 1 resumes the required geometrical parameters of the stirred system. The geometry of the classical anchor impeller, as well as those of the newly designed impellers, are highlighted in Figs. 1 and 2, respectively. Before beginning

| Table 1 |
|---------|
| Dimensions of the agitator. |
| D/H  | d/D | d/h | c/D | l_i/d | a/d | d_s/d |
|-------|-----|-----|-----|-------|-----|-------|
| 1     | 0.5 | 0.53 | 0.066 | 0.04  | 0.08 | 0.12  |
the investigations, mesh tests and validation with experimental data were performed (Table 2 and Fig. 3, respectively). The results of power consumption are given Table 3 and Fig. 7, while Figs. 5 and 6 illustrate the variation of axial velocity and shear rates, respectively. The same results of Fig. 7 are given in a tabulated form in Table 4.
Fig. 3. Validation of the results of power number \((N_p)\) vs. Reynolds number \((Re)\).

Table 4
Power number \((N_p)\) for different cases.

| \(Re\) | Case 1 | Case 2 | Case 3 |
|--------|--------|--------|--------|
| 10     | 10.06  | 10.29  | 10.5   |
| 20     | 5.03   | 5.26   | 5.39   |

2. Experimental Design, Materials and Methods

2.1. Materials

The performance of anchor-agitated tank is investigated in this paper. The standard setup consists of a flat-bottomed cylindrical tank with a diameter \(D\) and height \(H\) (Fig. 1). The tank is completely filled with (Glycerol) with a density \((\rho = 1.26\ \text{kg/m}^3)\) and a viscosity \((\mu = 1.49\ \text{kg/m s à 20 °C})\). The agitator presented in Fig. 1 is a standard anchor with two vertical blades of height \(h\) and width \(a\) fixed on the extremities to a horizontal arm to favor the vertical motion of the flow. Some modifications in the geometry of vessel and impellers are introduced. The dimensions of the agitator, as well as the different cases studied are presented in Figs. 1, 2 and Table 1.

2.2. Method of investigation

The CFD (Computational Fluid Dynamics) tool has been used to achieve the investigations. The software employed (CFX) is based on the finite volume method. The geometry and mesh of the computational domain were generated with the computer tool Ansys ICEM CFD. The computations were conducted under the following considerations: steady-state, incompressible fluid, three-dimensional, and laminar flows conditions. The Rotating Reference Frame (RRF) approach was employed in modeling of the rotating elements. This technique has been selected due to the absence of baffles [6–8]. To achieve the velocity-pressure coupling, a pressure-correction method of the type Semi-Implicit Method for Pressure-Linked Equations-Consistent (SIMPLEC) was used [9,10]. The grids were refined near the impeller and vessel walls to capture the flow boundary details. To determine the optimal grid size for the computational domain, the mesh density was increased by about 2. Mesh tests were carried out by checking that the additional grids did
not change the velocity magnitude in regions with high gradients by more than 2.5% (Table 2). From these results, the mesh M2 with a global number of cells of about 835,159 was selected as optimal.

2.3. Validation

To verify the appropriate setting of boundary conditions and computational grid elements, some predicted results were compared against the available experimental data. Fig. 3 summarizes a comparison between our results of power number and the experimental data of Prajapati and Ein-Mozaffari [8].

3. Data Obtained

After analysis of the velocity fields generated inside the tanks for both the flat- and curved-bottomed tanks (Fig. 4), it is clear that the well-stirred region is the largest for Case No. 1 compared to the standard geometry with an energy consumption reduced by 1.65% (Table 3).

A series of numerical simulations allowed us to deduce useful insights into the geometrical modifications that can enhance the flow circulation (Figs. 4 and 5) and shear rates (Fig. 6), especially in the area between the bottom of the tank and the inferior part of the agitator.

Case No. 3 gives satisfactory flow characteristics and a moderate energy consumption, as observed in Fig. 7.

4. Data Analysis

The data assembled regarding the method of investigation, validation with experimental studies, as well as the main findings are analyzed and discussed in Figs. 3-7 and Table 4. From
these results, it seems that the newly modifications in the classical anchor impellers are interesting. Among the suggested geometrical modification, Case No. 3 seems to be the most efficient, since it provides satisfactory flow characteristics and moderate power consumption.

**Transparency document. Supplementary Material**

Transparency data associated with this article (the CSV file of Fig. 5, i.e. the tabulated form of the figure) can be found in the online version at ... The model used in simulation (CFX file, the source file) is also available online.
Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships which have or could be perceived to have influenced the work reported in this article.

Supplementary materials

Supplementary material associated with this article can be found in the online version at doi:10.1016/j.dib.2021.107669.

CRediT Author Statement

Youcef Kamla: Conceptualization, Methodology, Software, Validation, Writing – original draft; Houari Ameur: Investigation, Data curation, Writing – review & editing, Supervision; Mohammed Ilies Arab: Visualization; Belalia Azeddine: Visualization, Visualization, Visualization.

References

[1] F. Maluta, G. Montante, A. Paglianti, Analysis of immiscible liquid-liquid mixing in stirred tanks by electrical resistance tomography, Chem. Eng. Sci. 227 (2020) 115898.
[2] Q. Kang, D. He, N. Zhao, X. Feng, J. Wang, Hydrodynamics in unbaffled liquid-solid stirred tanks with free surface studied by DEM-VOF method, Chem. Eng. J. 386 (2020) 122846.
[3] H. Ameur, Energy efficiency of different impellers in stirred tank reactors, Energy 93 (2015) 1980–1988.
[4] Y. Kamla, H. Ameur, A. Karas, M.I Arab, Performance of new designed anchor impellers in stirred tanks, Chem. Pap. 74 (2020) 779–785.
[5] H. Ameur, Y. Kamla, Geometrical modifications of the anchor impeller to enhance the overall performances in stirred tanks, Instal 6 (2020) 42–45.
[6] R. Alcamo, G. Micale, F. Grisafi, A. Brucato, M. Ciofalo, Large-eddy simulation of turbulent flow in an unbaffled stirred tank driven by a Rushton turbine, Chem. Eng. Sci. 60 (2005) 2303–2316.
[7] A. Khapre, B. Munshi, Numerical investigation of hydrodynamic behavior of shear thinning fluids in stirred tank, J. Taiwan Inst. Chem. Eng. 56 (2015) 16–27.
[8] P. Prajapati, F. Ein-Mozaffari, CFD investigation of the mixing of yield-pseudoplastic fluids with anchor impellers, Chem. Eng. Technol. 32 (2009) 1211–1218.
[9] H. Ameur, Modifications in the Rushton turbine for mixing viscoplastic fluids, J. Food Eng. 233 (2018) 117–125.
[10] H. Ameur, Effect of the shaft eccentricity and rotational direction on the mixing characteristics in cylindrical tank reactors, Chin. J. Chem. Eng. 24 (2016) 1647–1654 2016.