Journal of the Taiwan Institute of Chemical Engineers 000 (2015) 1–12

Contents lists available at ScienceDirect

Journal of the Taiwan Institute of Chemical Engineers

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

Numerical investigation of hydrodynamic behavior of shear thinning fluids in stirred tank

Akhilesh Khapre*, Basudeb Munshi

Department of Chemical Engineering, National Institute of Technology, Rourkela, Odisha 769008, India

ARTICLE INFO

Article history:
Received 19 January 2015
Revised 27 March 2015
Accepted 7 April 2015
Available online xxx

Keywords:
CFD
Shear thinning fluid
Stirred tank
Reynolds number similarity
Power and flow number
Entropy generation

ABSTRACT

A simulation study reported on the velocity fields and the entropy generation of non-Newtonian fluids in a baffled stirred tank with Rushton turbine impeller. Two shear thinning fluids, carboxymethylcellulose (CMC) and xanthan gum (XG), with flow index (n) varying in the range from 0.64 to 0.85 are used as the working fluid. The steady state multiple reference frame and the realizable k–ε turbulence model is used for numerical simulation. The predicted velocities for a CMC solution with $n = 0.85$ compared with literature data, and overall good agreement observed. The effect of flow index on Reynolds number similarity, power number and flow number also studied. The present work also determines the entropy generation due to the fluid flow. The CFD simulation is applied to predict the effect of size of the impeller blade, impeller clearance, fluid flow behavior index and rotations of impeller on the spatial distribution of the total entropy generation.

1. Introduction

The performance of mixing affects the quality of mixed product and conversion of reactant in a reactor. Non-Newtonian fluids are widely used in the oil, chemical and food industries. Viscosity is the key factor for the flow behavior of non-Newtonian fluid [1]. The required torque for achieving the satisfactory mixing becomes very high for highly viscous non-Newtonian fluid. It may lead to destruction of mixing system [2]. Hence, the mixing of highly viscous non-Newtonian fluid is usually carried out at low Reynolds number in laminar or the early transition regime [3]. Even for the low viscous non-Newtonian fluids, the study of mixing is restricted to laminar and transition regimes [4–8]. Nouri and Whitelaw [9] and Koutsakos and Nienow [10] investigated the effect of non-Newtonian liquid properties on the mixing performance of stirring vessels. Brito-De La Fuente et al. [3] studied the mixing of 10 different complex fluids using helical ribbon and helical ribbon screw impellers. They observed an inverse relation between the power consumption and pseudoplasticity. The hydrodynamic behavior of non-Newtonian fluid in the turbulent zone is seldom studied, may be due to operation difficulty in the turbulent zone. Venneker et al. [11] reported LDA (laser Doppler anemometry) measurements of the turbulent velocity fields in vessels agitated by a Rushton turbine. The working fluid had flow behavior indexes in the range of 0.56–1.0. They observed a Reynolds similarity [12] where flow behavior becomes independent of Reynolds number.

The collection of the experimental hydrodynamic data is difficult due to opacity of non-Newtonian fluids. Hence, the experimental study on the hydrodynamic behavior of the non-Newtonian fluid is quite less. Fortunately, the tools of the Computational Fluid Dynamics (CFD) help us to study the flow behavior of non-Newtonian fluids. Thibault and Tanguy [6] analyzed power drawn by a coaxial mixer in homogeneous Newtonian and non-Newtonian fluid in the laminar region. Kelly and Gigas [13] extended the study to transition flow regime. Shekhar and Jayanti [14] performed CFD simulations of pseudo plastic fluids of high viscosity in the laminar flow regime. Montante et al. [15] calculated the mixing time in a stirred tank agitated with 45° pitched blade turbines. Dular et al. [2] used six bladed vane rotors to predict different features such as vortex shapes, free surface shapes and torques on the shaft in non-Newtonian fluid mixing process. Arratia et al. [16] carried out both the computational and experimental works on the mixing of shear thinning yield stress fluids in a stirred tank with single and multiple impellers. Rudolph et al. [17] analyzed the power consumption in the laminar regime for an anchor impeller with a dual set pitched blade turbines. Minge et al. [18] carried out CFD simulations of non-Newtonian flow field generated in the laminar region by a double helical ribbon impeller. Um and Hanley [19] simulated bench scale bioreactor with a Rushton impeller to understand and improve mixing and mass transfer in a highly viscous non-Newtonian system. Wu [20–22] validated CFD models of mixing in anaerobic digesters with the liquid manure. Ameur and Bouzit [23] studied the effect of impeller speed, fluid rheology, and number of impeller blades on the induced flow patterns and the power consumption in a cylindrical unbaffled vessel at laminar and transition regime.

* Corresponding author. Tel.: +91 7894044869; fax: +91 7894044869.
E-mail address: akhilesh_khapre@yahoo.co.in (A. Khapre).

http://dx.doi.org/10.1016/j.jtice.2015.04.003
1876-1070/© 2015 Published by Elsevier B.V. on behalf of Taiwan Institute of Chemical Engineers.

Nomenclature

C_i	impeller off bottom clearance (m)
D	tank diameter (m)
Di	impeller diameter (m)
H	tank height (m)
K	consistency index (kg s^{n-2}/m)
n	flow behavior index (–)
N	impeller rotational speed (rps)
N_p	power number (–)
N_q	flow number (–)
r	radial distance (m)
R_i	impeller radius (m)
Re	impeller Reynolds number (–)
$\dot{S}_{gen}$	total entropy generation per unit volume (J/(s K) m^3)
$\dot{S}_{HT}$	entropy generation due to heat transfer per unit volume (J/(s K) m^3)
$\dot{S}_{VD}$	entropy generation due to viscous dissipation per unit volume (J/(s K) m^3)
$\dot{S}_{\overline{VD}}$	entropy generation due to mean viscous dissipation per unit volume (J/(s K) m^3)
$\dot{S}_{VD'}$	entropy generation due to fluctuating viscous dissipation per unit volume (J/(s K) m^3)
T	temperature (K)
V_{tip}	impeller tip velocity (m/s)
W_b	baffles width (m)
Greek letters	
μ	viscosity (Pa s)
η	apparent viscosity (Pa s)
μ_{eff}	effective viscosity (Pa s)
ρ	density (kg/m^3)
φ_v	viscous dissipation function (s^2)
ε	dissipation due to turbulence kinetic energy (m^2/s^3)
λ	thermal conductivity (W/(m K))

Entropy generation is an important parameter for studying the mixing process. In fluid flow, entropy is generated due to heat transfer and viscous dissipation of the fluid. For isothermal flows, the entropy generation consists of only viscous dissipation of fluid. Many researchers studied the rate of entropy generation experimentally and numerically in a various system with or without heat transfer process [24–31]. But only a few works on entropy generation in stirred tanks are available in the literature. Naterer and Adeyinka [29] studied experimentally entropy generation in laminar fluid motion induced by a magnetic stirrer in a rectangular tank using particle image velocimetry. They determined the entropy generation rate using the velocity gradients. Driss et al. [31] compared the experimental data with the CFD simulated results of a stirred tank reactor to show the effect of multiple (one, two and three) Rushton turbine configurations on the mixing performance.

Thorough literature survey showed that no one has predicted theoretically and numerically the Reynolds number similarity of non-Newtonian fluid in the turbulent zone. Less amount of theoretical study of the non-Newtonian fluid flow behavior in stirred tank with radial impeller was studied. The present paper discusses a number of issues which include the prediction of experimental hydrodynamics data available in the open literature [11,32–35] and also the prediction of Reynolds number similarity of non-Newtonian fluid. Beside these, the present work has used CFD models to predict and analyze many important observations of Venneker et al. [11]. In mixing process, entropy generation increases at the expense of additional power input to the impeller [29]. The effect of the system parameters like the flow index of fluid, the size of the impeller blade, impeller clearance and the rotational speed of the impeller on the entropy generation of the stirred tank are not studied previously. Therefore, this paper has also focused on the numerical analysis of entropy generation in the isothermal stirred tank due to the change of the system parameters.

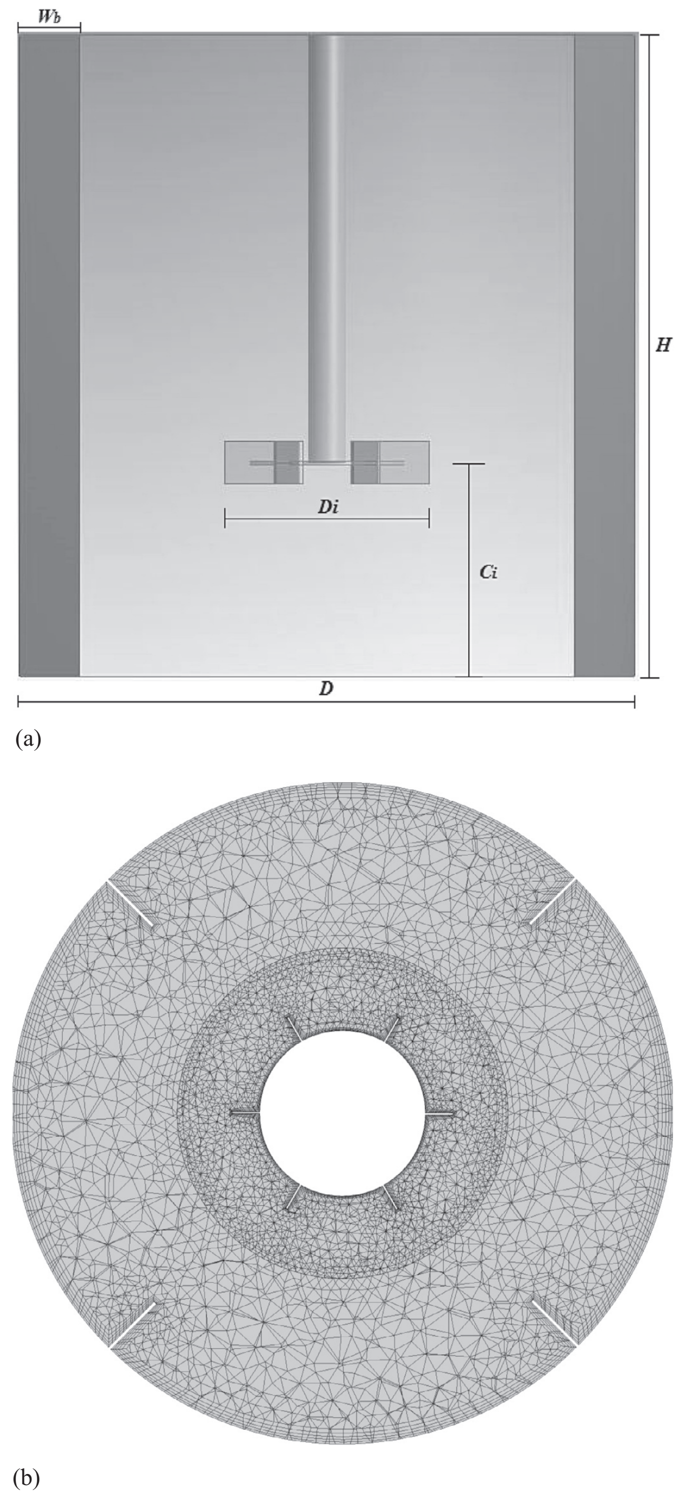

Fig. 1. (a) Geometry of stirred tank; (b) computational mesh (tetrahedral mesh).

2. Physical system

The simulation work needs a basic problem with available experimental data in the open literature. A number of choices are available for Newtonian fluid. But it gets restricted for non-Newtonian fluid. In the present study, the system is taken from Venneker et al. [11] and shown in Fig. 1. It represents a flat bottom baffled stirred tank with a