Computational Study of Mixing of Shear Thinning Fluids with Modifications in Rushton Turbine Impeller

Aishwarya Mulampaka and K. S. Rajmohan

Abstract Mixing is one of the most common and vital operations in industries such as the chemical, cement, food, and polymer industry. It is necessary to understand the hydrodynamics of the mixing tank and the mixing behavior of the fluids to assess the quality of mixing and the energy consumption to optimize the mixing process. From the literature survey, it is observed that there are plenty of experimental and computational studies done on Newtonian fluids whereas only limited studies related to non-Newtonian fluids have been reported. Generally, the impeller geometry is selected based on the viscosity of a fluid. The turbulence effect around the region of the impeller, pumping capacity, and power consumption can be best studied with the help of computational fluid dynamics. The simulation of the mixing process is done using ANSYS Fluent, where the velocity vector plots, contours, and streamlines can be studied and analyzed, which in turn will help to optimize the design. In the present research work, study related to various turbulence models and rotating approaches in the CFD, prediction of hydrodynamic behavior of Newtonian and non-Newtonian fluids is made to validate the model. Modifications in the Rushton turbine impeller are made by introducing cuts in the blade of the Rushton turbine impeller. A comparison of the results regarding power consumption and mixing time will be made to identify the optimum design with less power consumption and shorter mixing time for shear thinning fluids.

Keywords Shear thinning fluids · Computational fluid dynamics · ANSYS Fluent · Power consumption

---

A. Mulampaka · K. S. Rajmohan (✉)
Department of Chemical Engineering, National Institute of Technology Warangal, Warangal 506004, Telangana, India
e-mail: rajmohan@nitw.ac.in

© The Author(s), under exclusive license to Springer Nature Singapore Pte Ltd. 2022

R. P. Bharti and K. M. Gangawane (eds.), *Recent Trends in Fluid Dynamics Research*, Lecture Notes in Mechanical Engineering,
https://doi.org/10.1007/978-981-16-6928-6_1

Nomenclature

K	Consistency index (kg s^{n-2} /m)
n	Flow behavior index
ρ	Density (kg/m^3)
ε	Dissipation due to turbulence kinetic energy (m^2/s^3)
μ	Viscosity (Pa s)
D_i	Impeller diameter (m)
N	Impeller rotational speed (rps)
η	Apparent viscosity (Pa s)
Np	Power number
k	Specific turbulent kinetic energy(m^2/sec)
P	Power required for running agitator (W)
N_q	Flow number
ϕ_v	Viscous dissipation function (s^2)

1 Introduction

Mixing is one of the oldest and extremely important operations with countless applications in almost every chemical, food processing, and polymer industry. It involves blending of two immiscible fluids, homogenization, dispersion of solids into liquids, and gases in liquids. Based on the type of operation, they can be classified as batch, continuous, and semi-batch. It is important to study the hydrodynamics of the mixing tank and the mixing behavior to assess the quality of mixing and the energy consumption to optimize the mixing process. From the literature survey, it has been found that there are many experimental and computational studies done on Newtonian fluids and few studies are related to non-Newtonian fluids. Non-Newtonian fluids are of vital importance in the food and pharmaceutical industries. Based on the fluid properties, the impeller is selected. The mixing operation greatly depends on the impeller geometry, presence of baffles, type of tank, and presence of draft tubes. The turbulence effects around the region of the impeller can be best studied with the help of computational fluid dynamics. The simulation of the mixing process is be done using ANSYS Fluent, where the velocity profiles, vector, contour, and streamline plots can be studied and analyzed, through which the optimization of the design can be done. In the present research work, study related to different turbulence models and rotating approaches in the CFD, prediction of hydrodynamic behavior of Newtonian and non-Newtonian fluids is made to validate the model. Energy saving is the most concerning aspect in mixing operations.

Many studies have been done to study the effect of impeller geometry on power consumption [1]. The power draw depends greatly on the impeller blades, blade spacing, clearance, disc thickness, shaft inclination, and eccentricity [2]. Modifications in the Rushton turbine impeller made by Rao [3] are limited to Newtonian fluids. Both experimental and simulation work have been reflected. Based on this research work, Houari Ameur has adapted the modification in Rushton turbine blades to viscoelastic fluids from the literature data mentioned [4]. The studies show that the cavern size (well mixed) region is great for standard Rushton turbine but with additional power consumption when compared to blades with cuts [5]. In the present research work, the idea of modified blades from [3] has been adapted to test the effect of power consumption when shear thinning fluids (pseudoplastic) are being used. The velocity flow field, power number, and mixing time will be compared and the best design with less power consumption and shorter mixing time is predicted with the help of computational fluid dynamics. The effect of fluid flow and impeller characteristics on the hydrodynamic behavior of a continuous stirred tank reactor (CSTR) has been studied using the computational fluid dynamics (CFD) method [12]. A detailed description of tank geometry, baffles, impeller, and draft tubes are given with a description of experimental and computational techniques are provided with Classification on types of discretization schemes and modeling approaches [13].

2 Problem Statement

The geometry of the mixing tank is created using SpaceClaim, with standard dimensions. Data validation with shear thinning fluids is done with reference to [6]. The dimensions of the standard Rushton blade turbine are mentioned in Table 1. Modification in the blade design is done with reference to [3], and the dimensions are taken from the same reference, mentioned in Table 1. The main idea is to predict how the power consumption and mixing time vary when the cuts in the blade are introduced, which are depicted in Fig. 1. This can help in selecting the best design suitable for the mixing operation with less power consumption and shorter mixing time.

Table 1 Tank and blade dimensions

Tank	Dimensions (mm)
Tank height (H)	270
Tank diameter (T)	270
Impeller diameter (D = T/3)	90
Blade height (D/5)	18
Blade length (D/4)	22.5
Baffle width (T/10)	27
Baffle thickness (T/100)	3

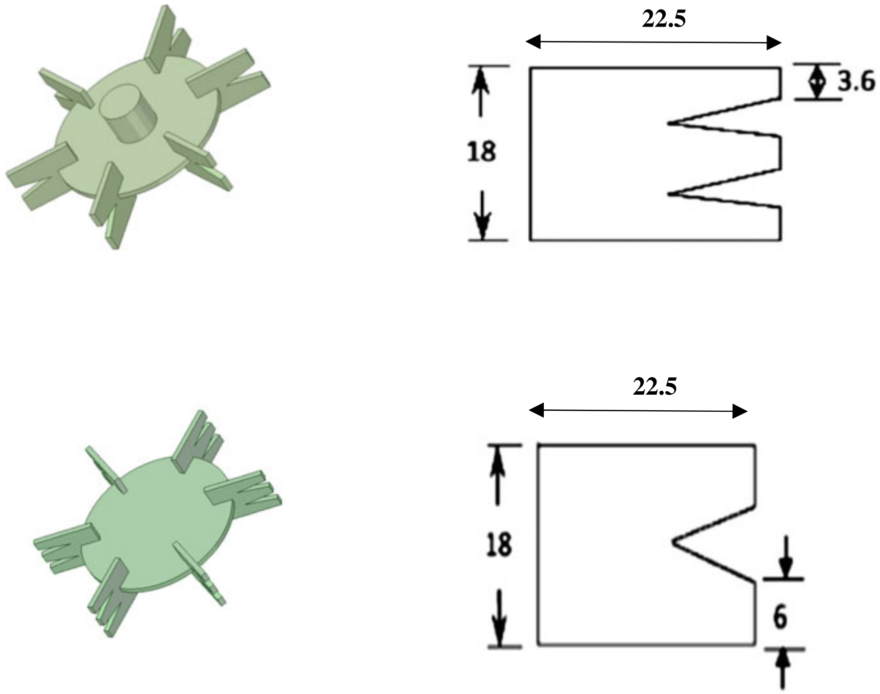

Fig. 1 V—cut turbine and W—cut turbine image with dimensions

2.1 Mathematical Model

A non-Newtonian fluid is a fluid that deviates from "Newton's law of viscosity." The viscosity of a non-Newtonian fluid is variable and may vary with stress, with time, or with a combination of both. Based on this behavior, the fluids are classified as pseudoplastic, thixotropic, rheopectic, and dilatant. The different non-Newtonian fluids as jam, butter, carboxymethyl cellulose, xanthan gum, sauces, yogurt, detergents, etc. Based on their viscous behavior, the fluids can be classified as follows [7]. The working fluids xanthan gum, carboxymethyl cellulose at different concentrations are taken with reference to [6]. The rheology of the working fluids is shown in Table 2.

Table 2 Rheological properties of working fluids

Working fluid	Wt%	Consistency index (K) [kg s^{n-2} /m]	Flow behavior index (n)
Carboxymethyl cellulose	0.1	13.2	0.85
Xanthan gum	0.045	9.5	0.8
Xanthan gum	0.08	34.0	0.64
Natrosol	1	10.8	0.59

2.2 Governing Equations

The commercial software, ANSYS Fluent, which uses a control volume technique to discretize the conservation equations, is used to solve the conservation of mass and momentum energy along with the other equations and to generate flow fields. The governing equations of continuity, momentum, and temperature are as follows [8].

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \overline{v}) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \overline{v}) + \nabla \cdot (\rho \overline{v}\overline{v}) = -\nabla P + \nabla \cdot \left(\overline{\overline{\tau}}\right) + \rho \overrightarrow{g} \tag{2}$$

$$\rho_{C_P} \frac{\partial T}{\partial t} + \rho C_p \nabla(\overline{v} T) = ke_{ff} \cdot \nabla^2 T - \phi_v \tag{3}$$

where $\overline{v}$ is velocity vector, T is temperature, P is the static pressure, $\overline{\overline{\tau}}$ is the stress tensor, $\overrightarrow{g}$ is the gravitational body force, K_{eff} is an effective thermal conductivity, C_P is the heat capacity of the liquid at constant pressure, and Φ_v is the viscous dissipation function.

The stress tensor $\overline{\overline{\tau}}$ is expressed as

$$\overline{\overline{\tau}} = \eta\left[\nabla \overline{v} + \nabla \overline{v}^T\right] - \frac{2}{3}\nabla \cdot \overline{v} I \tag{4}$$

where η is the apparent viscosity, I is the unit tensor.

The turbulent flow, which is induced by the Rushton turbine, is modeled by realizable k-ε turbulence model. The governing equations of turbulence kinetic energy, k, and its rate of dissipation, ε are

$$\nabla.(\rho k \overline{v}) = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla \cdot k\right] + G_k - \rho\varepsilon \tag{5}$$

$$\nabla \cdot (\rho \varepsilon \overline{v}) = \nabla \cdot \left[\left[\mu + \frac{\mu_t}{\sigma_\varepsilon}\right]\nabla \cdot \varepsilon\right] + C_2 \rho \frac{\varepsilon'}{k + \sqrt{\nu\varepsilon}} \tag{6}$$

in which C_2 is constant. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. The following values are used for the constants [8]

$C_2 = 1.92$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.2$.

In the equation, G_k represents the generation of turbulence kinetic energy due to mean velocity gradients and calculated as

$$G_k = -\rho \overline{u_i^{'} u_j^{'}} \frac{\partial u_j}{\partial x_i} \tag{7}$$

The turbulent or eddy viscosity μ_t is computed by combining k and ε as follows [8]

$$\mu_t = \rho c \mu \tfrac{k^2}{\varepsilon} = \mu_t s^2 \tag{8}$$

where S is the modulus of the mean rate of the strain tensor

$$S = \sqrt{2 S_{ij} S_{ij}},\ S_{ij} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x}\right)$$

The variable C_μ is calculated as

$$C_\mu = \frac{1}{A_0 + A_s \frac{kU^*}{\varepsilon}} \tag{9}$$

where $U^* = \sqrt{S_{ij} S_{ij} + \overline{\Omega}_{ij} \overline{\Omega}_{ij}};\ \overline{\Omega}_{ij} = \Omega_{ij} - 2\varepsilon_{ijk}\omega_k$

Ωij is the mean rate of rotation of tensor viewed in a moving reference frame with angular velocity ω_k. The model constants A_0 and A_s are given as

$\mathrm{A0} = 4.04$, $\mathrm{A_s} = \sqrt{6} cos\varnothing$

$$\phi = \frac{1}{3} cos^{-1}(6W),\ W = \frac{S_{ij} S_{jk} S_{ik}}{\overline{S}^3},\ \overline{S} = \sqrt{s_{ij} s_{ij}}$$

For a Newtonian fluid, the impeller Reynolds number for the stirred tank is given by

$$Re = \frac{\rho N D_i^2}{\mu} \tag{10}$$

where N is the rotational speed of the impeller, and ρ is the density of the fluid. For non-Newtonian fluid, the power law is used to model viscosity and is given as

$$K = \dot{\gamma}^{n-1} \tag{11}$$

in which η is apparent viscosity, K is consistency index, and n is the flow behavior index. The impeller Reynolds number for pseudoplastic fluid is calculated using the Metzner-Otto method [9]

$$\mathrm{Re} = \frac{\rho N^{2-n} D_i^2}{K \cdot k_s^{n-1}} \tag{12}$$

where k_s is Metzner-Otto constant with $k_s = 11.5$.

The power number of impellers is calculated by

$$N_p = \frac{P}{\rho N^3 D_i^5} \tag{13}$$

where P is the power input, which is calculated from the torque, Γ applied on the impeller shaft. Mathematically, calculated as

$$P = 2\Pi N\Gamma$$

3 Numerical Method

The three-dimensional flow of the shear thinning fluids in the mixing tank with different impellers is simulated using ANSYS workbench 17.2. This computer tool uses the finite volume method to solve the momentum and energy equations. The geometry is created using SpaceClaim CAD tools, and the computational domain is discretized with tetrahedral mesh as shown in Fig. 2. A grid independence test is conducted to check if the number of nodes in the mesh affects the velocity magnitude. The results are listed in Table 3. The second order upwind scheme has been used to

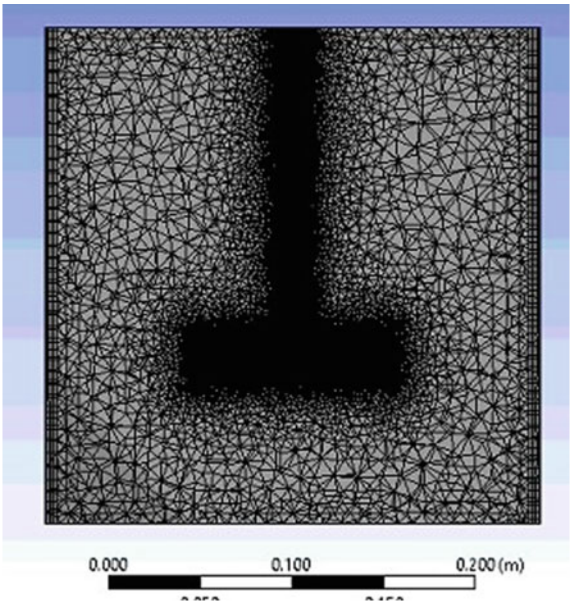

Fig. 2 Magnified view of mesh of the mixing tank

Table 3 Grid independence test

Element size (m)	No. of nodes	No. of Elements	Velocity (m/s)
4e-003 (fine)	475,647	2,325,578	0.960531
5e-003 (medium)	278,179	1,331,897	0.958653
6e-003 (coarse)	188,757	895,474	0.960149

discretize the convective terms in the momentum equations. A SIMPLE algorithm was used for solving pressure–velocity coupling.

In this study, we are using the standard k-ε model, it is the most commonly used model, it is robust and with less computational cost and has been useful in the engineering community for many years. It gives stable calculation and very suitable, especially for high Reynolds numbers.

Multiple reference frames (MRFs) model modified form of the rotating frame model uses several rotating and non-rotating frames. In the MRF approach, for the rotating frame, the impeller does not move. In a stationary frame with tank walls and baffles, the wall and baffles do not move. The rotating frame is under motion.

The non-Newtonian power-law model is activated by writing console code, and the materials are created with the corresponding rheological properties as mentioned in Table 2. Relative convergence criteria of 10–6 for the continuity and x- and y-components of velocity are defined.

4 Validation

Computational fluid dynamics is a multi-disciplinary subject that deals with fluid mechanics, numerical analysis, and data structures. It is a tool used to solve the conservation equations for mass, momentum, and energy. These equations are in the form of partial differential equations, which is an extremely tough task to solve analytically. CFD discretizes these equations from nonlinear termed equations to linear algebraic form, which are further solved to get accurate results for the corresponding fluid domain [10]. During the theoretical study of the subject, assumptions are made that the fluid is in the laminar region but most of the industrial processes deal with the turbulent conditions of fluid. Hence, it is necessary to study the effect of turbulence on the mixing phenomena for better understanding. Many studies have been done with various impellers to observe the power consumption and flow field inside the mixing tank system. The effect of fluid flow and impeller characteristics on the hydrodynamic behavior of a continuous stirred tank reactor (CSTR) has been studied using the computational fluid dynamics (CFD) method [12].

The geometric model is first validated with the literature date available from [11] for the same fluids as taken by the authors. Khapre and Munshi [6] used the same

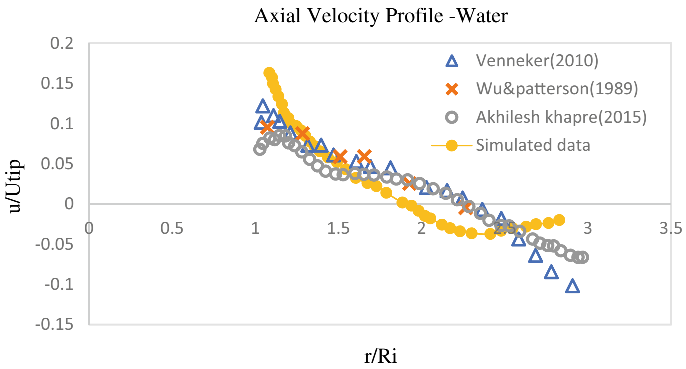

Fig. 3 Axial velocity versus normalized radial distance for water

system for validation but had conducted studies on entropy generation for different impellers and to study the effect of entropy with respect to the blade width. The comparison of the predicted results with the experimental data, Fig. 3, for water as working fluid shows good agreement.

5 Results and Discussion

5.1 Effect on Power Consumed

Every impeller has a unique power curve. From the literature, data are verified that the power curve obtained by simulation is in agreement with the unique power curve of the Rushton turbine impeller. The power consumption is highest in the laminar region and decreases linearly with the Reynolds number, showing the system is in agreement with unique power curve. The hydrodynamics and mixing behavior are largely affected by the impeller design and fluid properties. Power consumption is one of the hugely concerned factors. To minimize the power consumption and to study the effect of blade design, cuts have been introduced into the standard Rushton blade turbine impeller. The power curves for standard Rushton blade turbine, V-cut turbine, and W-cut turbine are shown in Fig. 4a–c at different rotational speeds, for different shear thinning fluids.

5.2 Effect on Mixing Time

Mixing time or blend time is one of the significant parameters to characterize the performance of the mixing tank. It is the time taken to achieve the maximum (99%)

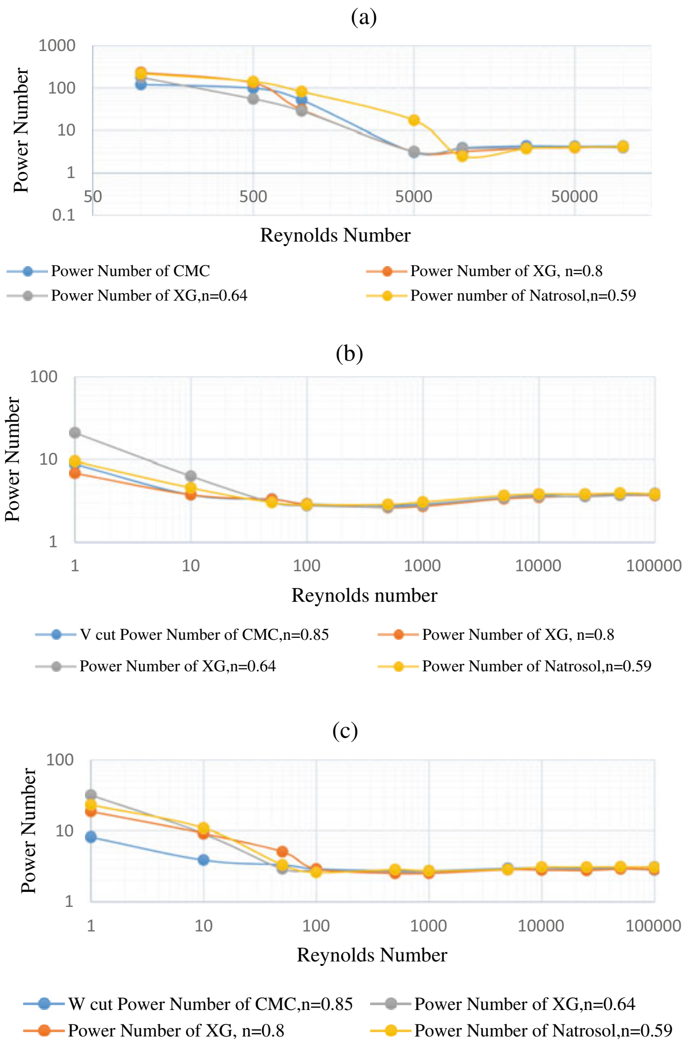

Fig. 4 The power curve for **a** standard Rushton blade turbine, **b** V-cut turbine, **c** W-cut turbine

of the steady-state concentration. The transport of a tracer helps to understand the degree of homogeneity in the agitated tank. Mixing time was predicted using "Transient transport of a neutrally-buoyant tracer (scalar)." The probe locations and tracer injection points are shown in Fig. 5.

The overall mixing time for standard design turbine is observed to be around 12 s as shown in Fig. 6a.

The mixing time for V-cut and W-cut turbines is observed to be around 9 s and 8 s, respectively, as shown in Fig. 6b, c.

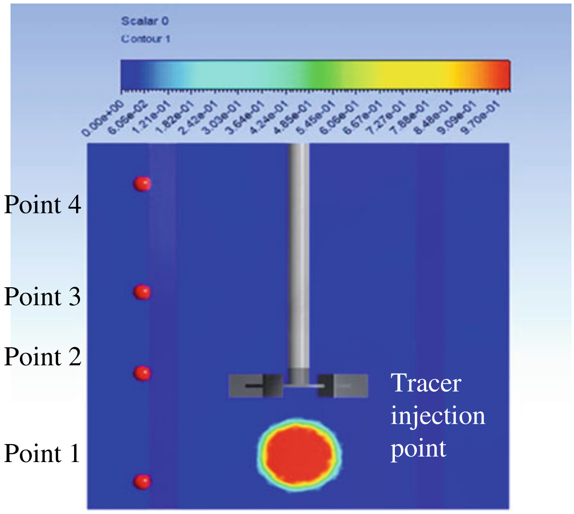

Fig. 5 Tracer injection and probe points inside the tank

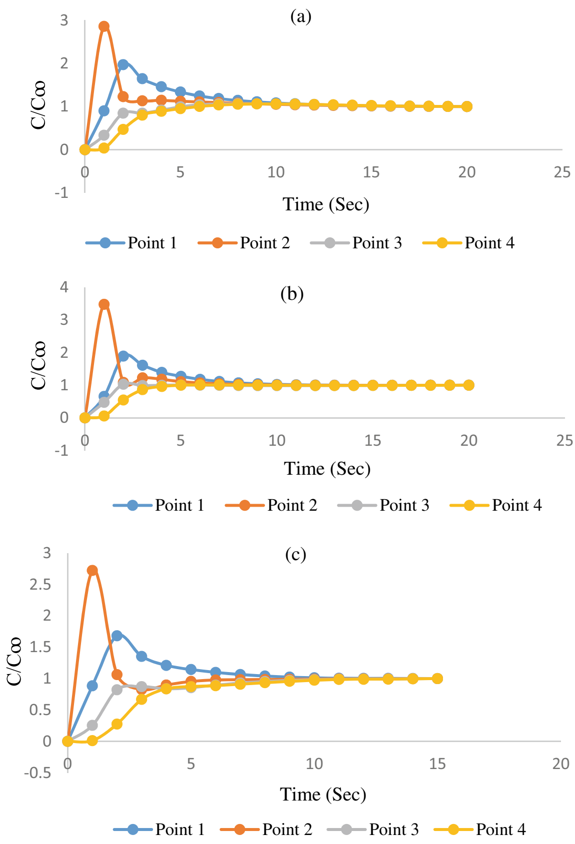

Fig. 6 Response curve of CMC at 180 rpm at point 1, point 2, point 3, and point 4 for **a** Standard Rushton blade turbine, **b** V-cut turbine, **c** W-cut turbine

The highest mixing time is observed for the standard Rushton blade turbine, followed by W-cut blade turbine, and the least time is taken by the V-cut blade turbine.

The reason for this order could be the strong impinging flow of fluid by the V-cut blade turbine allows the flow radially with stronger recirculation loops that are reaching the top surface of the tank. Though standard Rushton blade turbine has better radial flow with maximum cavern size, the time taken to reach the homogeneity is greater. Though the results of mixing time are differed by the value of one to two seconds, to find the best compromise among the mixing time power consumption, the studies of mixing time are considered here.

5.3 Effect on Cavern Size

In the food, polymer and pharmaceutical industries intimate contact between the fluid particles is required to achieve high-quality final products. This can be accomplished with an impeller that allows maximum contact of fluid in the stirred tank. To observe the contact area around the impeller, the cavern region (the well-mixed region around the impeller blades) is observed with the help of velocity contour plots along with the height of the tank. From Fig. 7a–c, it is evident that maximum cavern size is achieved by standard. Rushton blade turbine and V-cut turbine when compared to and W-cut turbine impeller. The maximum contact area is given by standard Rushton blade turbine followed by V-cut turbine, W-cut turbine.

6 Conclusions

- The importance of mixing in the chemical industries is highlighted. A geometric model has been created using ANSYS Fluent.
- The simulated results for both Newtonian and non-Newtonian fluids have good agreement with the literature data. Grid independence test has been conducted, and it is proven that the results are independent of the mesh size.
- The power number results for standard design and modified (V-cut design and W-cut design) are observed. It shows that the standard Rushton turbine has consumed power of 10.7% more than that of V-cut turbine. When compared with the W-cut turbine, it consumed 39% more power.
- Based on the mixing time observations, it is found that for standard design, V-cut, and W-cut turbines and it is 12 s, 10 s, and 13 s, respectively.
- The contour plots along the tank height show the cavern size to predict the maximum mixing region, and it is found that standard Rushton blade turbine gives the maximum cavern size followed by V-cut, W-cut.

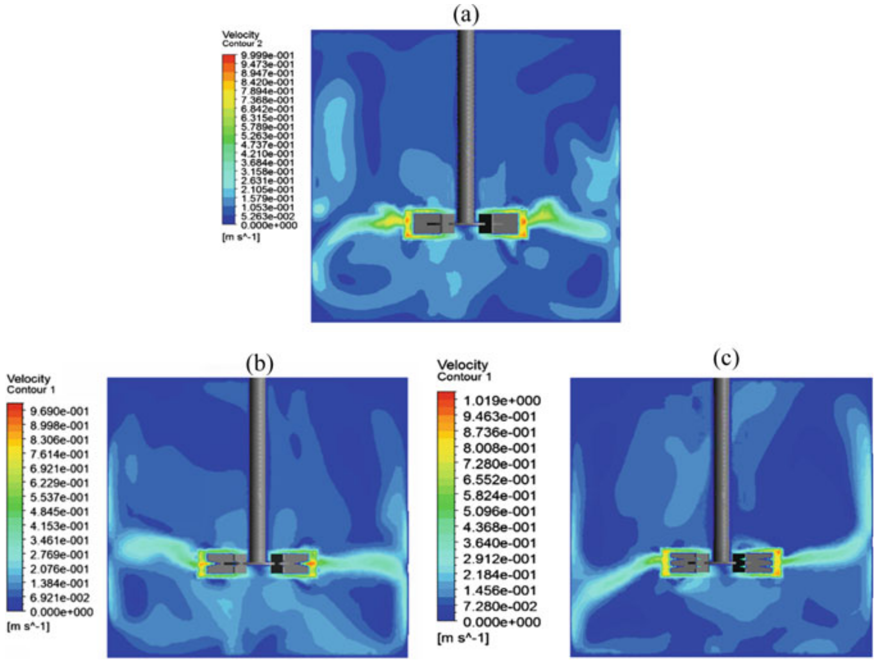

Fig. 7 Velocity contour plots of **a** Standard Rushton blade, **b** V-cut turbine, **c** W-cut blade turbine

- A compromise on both power consumption, cavern size, and mixing time for shear thinning fluid, CMC gives us the following order of preference for efficient mixing operation standard Rushton turbine > V-cut turbine > W-cut turbine.

References

1. Taghavi, M., Zadghaffari, R., Moghaddas, J., Moghaddas, Y.: Experimental and CFD investigation of power consumption in a dual Rushton turbine stirred tank. Chem. Eng. Res. Des. **89**(3), 280–290 (2011)
2. Wang, S., Wu, J., Ohmura, N.: Inclined-shaft agitation for improved viscous mixing. Ind. Eng. Chem. Res. **52**(33), 11741–11751 (2013)
3. Rao, D.A., Sivashanmugam, P.: Experimental and CFD simulation studies on power consumption in mixing using energy saving turbine agitator. J. Ind. Eng. Chem. **16**(1), 157–161 (2010)
4. Cortada-Garcia, M., Dore, V., Mazzei, L., Angeli, P.: Experimental and CFD studies of power consumption in the agitation of highly viscous shear thinning fluids. Chem. Eng. Res. Des. **1**(119), 171–182 (2017)
5. Ameur, H.: Modifications in the Rushton turbine for mixing viscoplastic fluids. J. Food Eng. **1**(233), 117–125 (2018)
6. Khapre, A., Munshi, B.: Numerical investigation of hydrodynamic behavior of shear thinning fluids in stirred tank. J. Taiwan Inst. Chem. Eng. **1**(56), 16–27 (2015)

7. Tavlarides, L.L., Stamatoudis, M.: The analysis of interphase reactions and mass transfer in liquid-liquid dispersions. In: Advances in Chemical Engineering, vol. 11, pp. 199–273. Academic Press (1981)
8. Ansys Fluent 13: User's Guide (2011). U.S.A.: Ansys Inc.
9. Metzner, A.B., Otto, R.E.: Agitation of non-Newtonian fluids. AIChE J. **3**(1), 3–10 (1957)
10. Ferziger, J.H., Perić, M., Street, R.L.: Computational methods for fluid dynamics. Springer, Berlin (2002)
11. Venneker, B.C., Derksen, J.J., Van den Akker, H.E.: Turbulent flow of shear-thinning liquids in stirred tanks—the effects of Reynolds number and flow index. Chem. Eng. Res. Des. **88**(7), 827–843 (2010)
12. Rajavathsavai, D., Khapre, A., Munshi, B.: Study of hydrodynamic behaviour of a CSTR using CFD. Conference Paper · March 2011. https://doi.org/10.13140/RG.2.1.2270.8161/1
13. Ochieng, A., Onyango, M., Kiriamiti, K.: Experimental measurement and computational fluid dynamics simulation of mixing in a stirred tank: a review. S. Afr. J. Sci. **105**(11–12), 421–426 (2009)