CHEMICAL ENGINEERING TRANSACTIONS

VOL. 111, 2024

A publication of AIDIC

The Italian Association
of Chemical Engineering
Online at www.cetjournal.it

Guest Editors: Valerio Cozzani, Bruno Fabiano, Genserik Reniers

ISBN 979-12-81206-11-3; **ISSN** 2283-9216

DOI: 10.3303/CET24111106

Computational Fluid Dynamics (CFD) Studies to Study the Effect of Liquid Level on the Hydrodynamics in an Agitated Vessel with an Axial and Radial Impeller

Kanad V. Patel, Dhruv T. Jagani, Arijit A. Ganguli*, Shuja Ahmed

School of Engineering and Applied Sciences, Ahmedabad University, India
arijit.ganguli@ahduni.edu.in

Hydrodynamics in stirred vessels is vital for transport phenomena for various applications in chemical and environmental systems like aerobic and anaerobic systems, effluent treatment and biological applications like bioreactors. Design of efficient impellers reduces power consumption which contributes to reduction in greenhouse gases especially power obtained from thermal power plants thus decreasing the carbon footprint. In the present work, Computational Fluid Dynamics (CFD) simulations have been carried out for two different Rushton turbines (one axial and one radial flow) for two different height to diameter (H/T) ratio. The model is first validated for a Pitched Blade Turbine Downflow (PBTD) impeller with data available in the literature. A qualitative study of the flow patterns in stirred tanks shows good axial and radial flow profiles for the respective impellers. A decrease in H/T showed higher power consumption for both impellers. The power number (N_p) for these impellers has also been calculated for two H/T ratio and Reynolds number (Re) in the range (2500 < Re < 1000. N_p was found to be a strong function of liquid level for both impellers considered in the study. Interesting trends of variation of N_p with Re have been observed which suggest there is a critical Re where the impellers have least power consumption

1. Introduction

Understanding of the fluid dynamics inside reactors has a significant effect on the performance of the reactor. With environmental sustainability gaining prime importance, Improving impeller designs for reducing power consumption both numerically and experimentally has been carried out since more than 4 decades. However, geometric and operating parameters also play a vital role in the amount of power being consumed for the same impeller design. Kumaresan and Joshi (2006) have carried out experimental and simulation based studies on 15 radial and axial impellers to understand the flow patterns and radial velocity profiles for different axial positions. The authors also have performed energy balance for understanding the power consumption per unit volume for all cases along with an analysis on power number and mixing time. In both cases, the studies were limited to quantitative analysis without the qualitative aspects like velocity vectors, energy dissipation profiles and kinetic energy profiles. Basavarajappa et al. (2015) have carried out extensive CFD studies for different impeller types with different geometric variations. The authors observed drop in the power number in axial flow impellers for lower clearance and liquid level values. Devarajulu and Loganathan (2016) have carried out extensive experimentation with five radial and four axial type impellers with the objective of understanding effect of liquid level and impeller clearance on the critical impeller speed for solid suspension. The authors based on their experimental investigations found a new correlation for Zwietering constant for different height to width and clearance to width ratios. The authors have also related their analysis to the power consumption required by the impellers for various cases. Tamburini et al. (2019) carried out CFD studies for a radial flow impeller to understand the difference between baffled and unbaffled tanks. The authors found that predictions for unbaffled tanks were difficult to compare with experimental data with CFD analysis. Foukrach et al. (2020) have also carried out experimental studies on hydrodynamics of different impellers and designed novel impellers to reduce power consumption and improve economy.

Paper Received: 31 January 2024; Revised: 2 June 2024; Accepted: 7 August 2024
Please cite this article as: Patel K.V., Jagani D.T., Ganguli A.A., Ahmed S., 2024, Computational Fluid Dynamics (CFD) Studies to Study the Effect of Liquid Level on the Hydrodynamics in an Agitated Vessel with an Axial and Radial Impeller, Chemical Engineering Transactions, 111, 631-636 DOI:10.3303/CET24111106

Recently, Alberini et al. (2024) have performed experimentation in stirred vessels to study the power consumption for a different speeds at different liquid levels. The authors have also found a correlation between power number and velocity profiles which enhance the understanding of the system. In the present work CFD investigation of two novel impeller designs, comprising Rushton Turbine and Rushton 45, has been performed. The objectives of the present work are: (a) Development and validation of a CFD model for the above mentioned axial and radial impeller and validation with experimental data of Kumaresan and Joshi (2006). (b) Perform a qualitative analysis of flow patterns for each impeller for two different H/T's. (c) Estimate the power number for different Re and two height to diameter (H/T) ratio's for both impellers considered.

2. Geometric details and Methodology

2.1 Geometry, operating parameters and grid sensitivity

The impeller geometry, the tank, baffles and both the impellers has been taken as per the data of Devarajulu and Loganathan (2016). The impeller speed is fixed at 630 rpm for the simulations. The placement of the impeller is as such, that the distance between the bottom of the impeller and to the bottom of the vessel is T/5.8. The geometry of PBTD 3020W50-6 (Pitched Blade Turbine Downflow with angle 30 degree at top and 20 degree bottom with width 50 mm and 6 blades) is shown in Figure 1A. The locations have 1 and 2 for z/H have been shown in Figure 1A. Only 60% of the tank is shown hence, the line 3 depicting z/H of 0.8 is not shown. These locations have been used for the velocity profiles in section 3.2.3

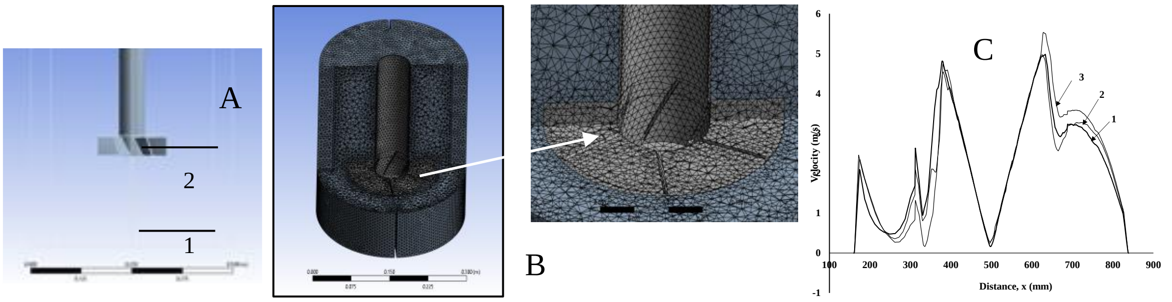

Figure 1 (A) Zoomed view of geometry with impeller and 50% of the tank (from bottom to centreline) 1. Position for of radial velocity profile near the bottom 2. Position for radial velocity profile near the impeller (B) Meshed geometry of PBTD3020W50-6, 628,000 no. of nodes (C) Grid sensitivity Mesh numbers 1. 514000 2. 467000 3. 850000

The k-epsilon turbulence model in a single-phase flow and turbulent conditions as per the experimental conditions reported by Kumaresan and Joshi (2006) and their simulations. The non-uniform mesh has been showed in Figure 1B. The deviation between the velocity profiles shown by a mesh size of 514000 and 850000 is found to be around 2% (as shown in Figure 1C) and hence the grid size of 514000 is chosen.

2.2 Methodology

A Pressure Staggered Option Scheme (PRESTO scheme) was used for pressure-velocity coupling, and second order upwind scheme was used for the momentum and turbulence energy equations. The residuals for pressure were kept 0.001 while for momentum and turbulence quantities was kept as 1E-05. The k-epsilon turbulence model, was used to capture the turbulent flow characteristics All simulations were carried out in an Intel I7 processor with 8 cores and 2 GHz processor speed.

3. Results and discussion

3.1 Validation

The validation of our computational model was based in the simulation of a stirred vessel as depicted in Figure 2. The geometrical dimensions for the tank, impeller, type of the impeller chosen for validation is the same as that of Kumaresan and Joshi (2006) as described in the previous section.

The dimensionless velocity versus dimensionless horizontal distance plot shows very good agreement away from the impeller while it shows a deviation of 7-9% for the flow near the impeller. Similar observations have been reported by Kumaresan and Joshi (2006) who have performed CFD simulations for the same impeller under same operating conditions. Hence, the validated CFD model has been chosen for the investigation of the newer designs of one axial and one radial flow impeller.

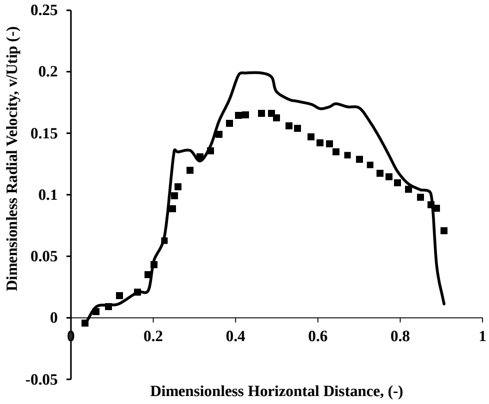

Figure 2 Comparison of radial velocities experimental versus CFD simulations. Symbols denote experimental values while line denotes CFD simulation

3.2 Qualitative and Quantitative Analysis

3.2.1 Flow patterns: Velocity vectors

All geometrical dimensions for the simulations are exactly equal to those of Devarajulu and Loganathan (2016). The major dimensions are reproduced namely the tank diameter which is 290 mm while the impeller diameter which is 96.7 mm.

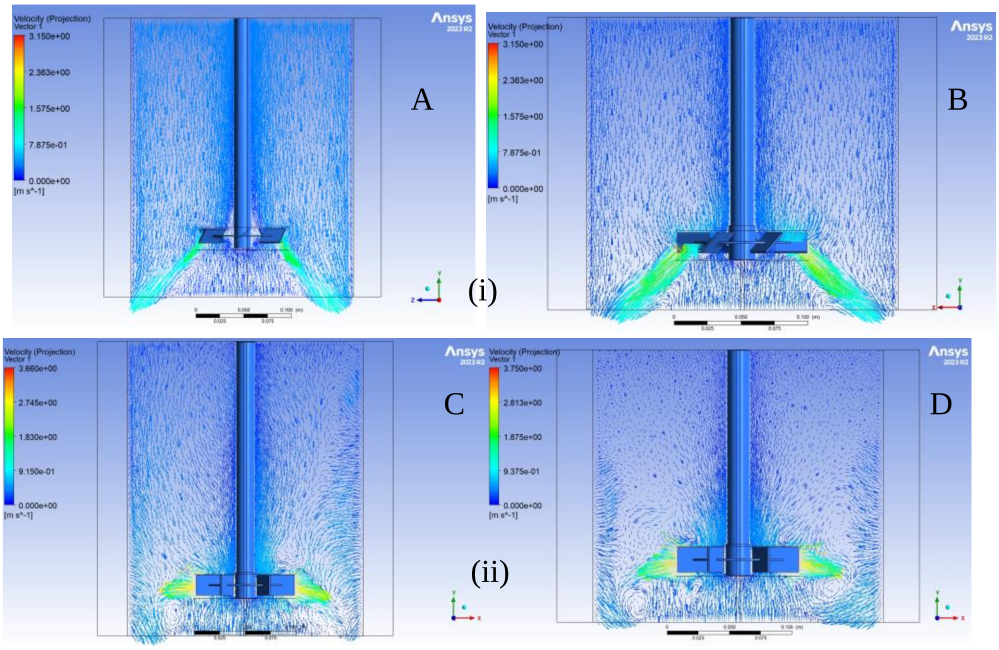

Figure 2 Velocity vectors (i) Rushton Turbine 45 (RT45) (A) H/T 1 and (B) H/T = 0.75 (ii) Velocity vectors for Rushton Turbine (RT) (C) H/T = 1 and (D) H/T = 0.75

Figure 3 (i) A and B show the flow patterns for two different water levels (H/T ratios) namely H/T=0.75 and H/T=1. Intense down flow from the tip of the blades can be observed for both H/T ratios considered. Further, the decrease in water level (H/T=0.75) causes a stronger down flow (with velocity magnitudes 7-8% higher) of the liquid as compared to H/T=1. A positive impact would be improvement in the bottom liquid circulations at a higher power consumption. Similar observations have been reported in the literature Rielly et al. (2007) where the authors reported higher power consumption for lower water levels as compared to those for same height

to diameter ratios. However, the strength of the two circulation cells (which is the circulation of the fluid from bottom to top on both sides of the shaft) are higher in case of H/T=1 than for H/T = 0.75. A closer look into the velocity magnitudes show that for H/T=0.75 the velocities are around 0.5 m/s in the part above the impeller (major portion of the tank) while corresponding values for H/T=1 were 0.65 to 0.7 m/s in the major part of the tank. These observations imply a more uniform mix and potentially reduced shear regions, which can be critical for sensitive biochemical processes.
Figure 3 (ii) (C) and (D) shows the velocity vectors for the Rushton Turbine at H/T ratios of 0.75 and 1 shows radial flow near the impeller implying efficient but potentially shear-intensive mixing. Further, secondary flows in the form of circulations near the tank bottom corners are observed in both the H/T's considered. The mixing as per visual observations seem to be higher in the case of lower H/T of 0.75. Further, the vectors, corresponding to an H/T of 1, depicts different hydrodynamics. Here, the velocity magnitude at the impeller blades is marginally reduced, with peak values observed near 3.0 m/s. The transition from high to intermediate velocities is smoother and extends further toward the tank walls. This indicates a more evenly distributed kinetic energy transfer throughout the vessel, which could be associated with a more homogenized mixing pattern. The reduced peak velocity also points towards a potentially more energy-efficient operation at full tank height. The effect on power consumption would be explained in the upcoming subsection..

3.2.2 Flow patterns: Turbulent Energy dissipation contours

Figure 4(i) A and B shows the turbulent energy dissipation contours for Rushton Turbine (RT) and H/T = 0.75 and 1 respectively while Figure 4(ii) C and D are contours for RT45.

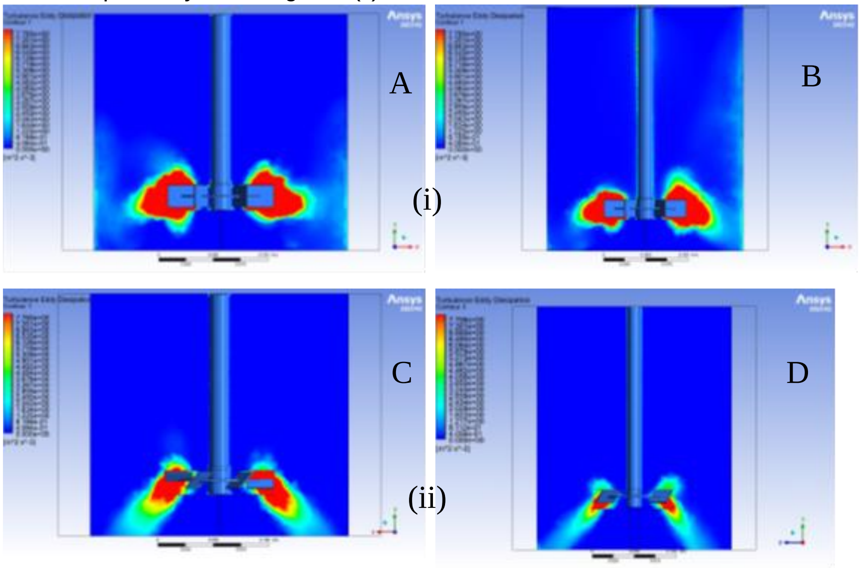

Figure 4 Turbulent Energy dissipation contours for Rushton Turbine 45^{0} blade angle (RT45) for (A) H/T = 0.75 and (B) H/T =1 (ii) Rushton Turbine (RT) for (A) H/T = 0.75 and (B) H/T =1

As can be observed the maximum amount of energy is dissipated in the vicinity of the impeller. Analysis of RT for the two H/T considered showed that the power consumed by the impeller (for the speed considered 630 rpm) was 5% higher for H/T = 0.75 than for H/T = 1. The contours of Figure 4(i) A and B also confirm that the area of energy dissipation near the impeller is slightly higher for H/T=1 and the volume average would result in higher values of energy dissipation and in turn power consumption. The analysis for turbulent energy dissipation contours for axial flow impeller (RT45) have also been shown in Figures 4 (ii) C and D. Similar results of power consumption have been observed for the liquid level for RT45 with higher magnitudes. The power consumption for liquid level H/T=0.75 was observed to be 7% higher than that of H/T=1. Qualitative contours of turbulent energy dissipation also depict higher values of energy dissipation in the bottom region of the tank for a low H/T ratio as compared to the high H/T. This higher area area of dissipation is sharply delineated, which tapers off rapidly as the flow moves axially downwards, signifying a strong shear region conducive to breaking down macro structures and promoting initial mixing.
At an H/T ratio of 0.75, the velocity contour showcases a well-defined high-velocity zone adjacent to the impeller blades, indicative of a vigorous radial discharge. The blade tips exhibit the highest velocities, 3.04 m/s, suggesting an intensive energy input at the impeller vicinity. The lower H/T ratio is characterized by a more intense, less uniform, radial flow which is beneficial for applications requiring high shear. In contrast, the

higher H/T ratio results in a more consistent and gentler flow pattern, which might be preferable for processes sensitive to shear stress.

3.2.3 Radial velocity profiles

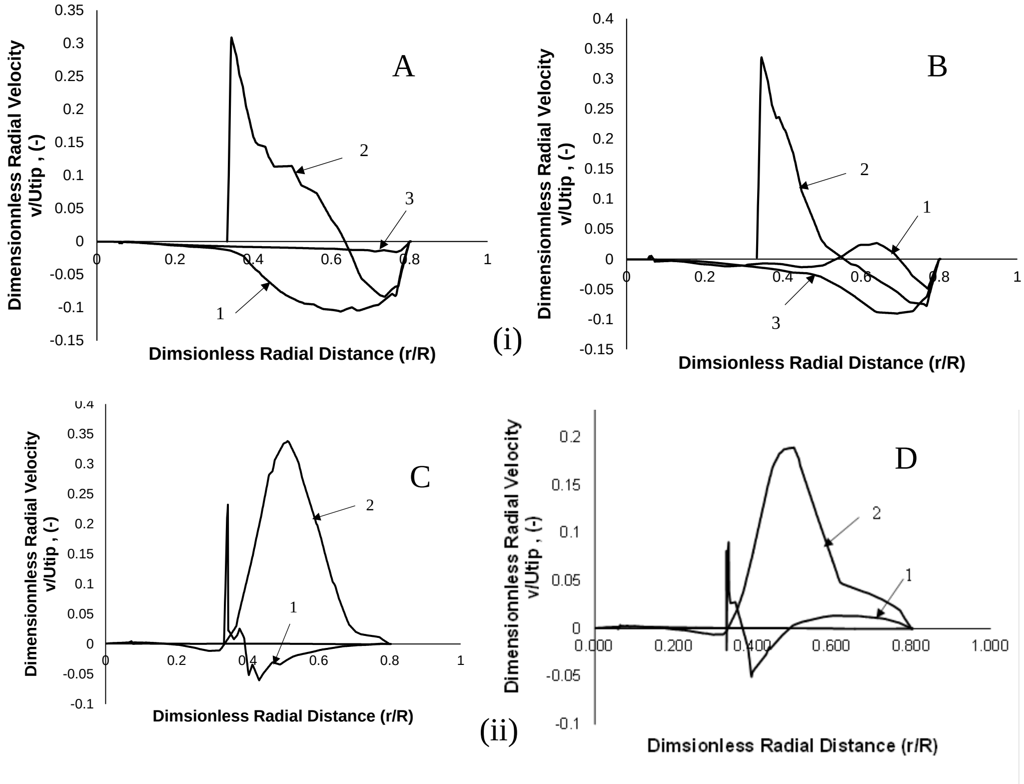

Figure 3 Turbulent Energy dissipation contours for (i) Rushton Turbine (RT) for (A) H/T = 0.75 and (B) H/T =1.(ii) Rushton Turbine 45⁰ blade angle (RT45) for (A) H/T = 0.75 and (B) H/T =1 1 z/H = 0.05; z/H = 0.3; 3 z/H = 0.8

The investigation of radial velocity profiles has been undertaken to understand hydrodynamics at three different levels of the tank. Figures 5 A – D show the radial velocity profile variations for both RT and RT45 impellers. Three different axial positions have been chosen for the analysis namely z/H = 0.05, 0.3 and 0.8. For RT impeller upflow velocities are thrice the downflow velocities near the impeller in both water levels. Due to this reason the velocities at the bottom of the tank are spread well indicating good liquid circulation and mixing. Similar liquid circulation can also occurs near the top level with velocity ratios 6 to 7 times lower than the maximum dimensionless velocity. Profiles of the radial impeller however shows only positive radial flow near the impeller with nonuniform liquid circulation at the bottom of the tank for both liquid levels investigated. Negligible or no flow is observed at the top of the tank

3.3 Power number evaluation

Power number (N_p) has been evaluated for all cases using the equation below:

$N_P = \frac{\int_0^R \int_0^H \int_0^{2\pi} \rho \varepsilon r dr dz d\theta}{\rho N^3 D^5}$ (Refer Nomenclature for variable definition) (1)

Figures 6A and B shows the variation of Np with Re for RT and RT 45 respectively, for two different values of H/T. Figure 6A shows Np for RT for H/T = 0.75 varied from 3 to 1.6 for 2500 < Re < 9700 with a minima at Re = 6500 while the one for H/T=1 varied from 2 to 1.5 for 2500 < Re < 9700 with a minima at Re = 6500. Correspondingly for RT45 Figure 6B shows that for H/T = 0.75 a different trend is observed. The N_p values were constant at 0.4 for 2500 < Re< 7000 after which a steep increase in N_p was observed upto 0.55. For H/T=1 a reverse trend is observed with N_p decreasing linearly from 0.5 to 0.3 for 2500 < Re< 7000 after which it remains constant at N_p = 0.3. This confirms that lower liquid levels (H/T=0.75) have higher power consumption than H/T=1 for all speeds except for RT45 where Np values are higher for Re range 2500 < Re < 4750.

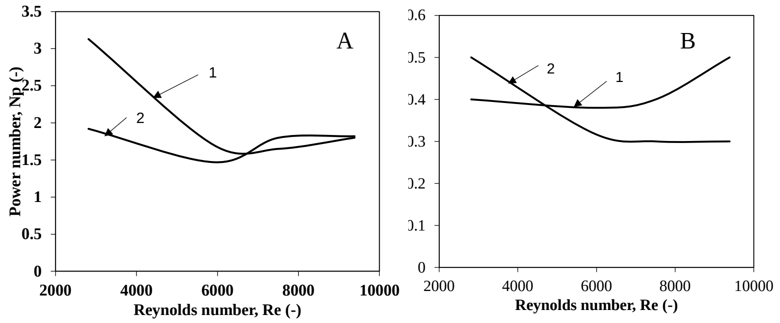

Figure 6 Variation of N_p with Re (A) RT (B) RT45 1. H/T = 0.75 2. H/T = 1.

4. Conclusion

Conclusions regarding the hydrodynamics and power consumption for the axial and radial impeller selected for study are:

1. The liquid level has considerable effect on hydrodynamics in baffled stirred vessels. Both axial flow (RT45) and radial flow (RT) impeller show stronger higher velocities for lower H/T.
2. Turbulence Energy dissipation is highest near the vicinity of the blades and higher power consumption happens for lower H/T
3. The power number (N_p) shows interesting trends for variation with Re. Lower H/T has higher Np values for both impellers except for lower Re in the range (2500 < Re < 5000) where Np values are lower for H/T=0.75 than that of H/T=1.
4. Np undergoes a minima for RT impeller at 6200.
5. The radial impeller has at least 3 times low power consumption than axial impeller for all simulated cases

Nomenclature

H – Water Height, (m); T – Tank Diameter, (m); R – radius of the tank (m); r – variable for change in radius (m); z = variable for differential height (m); θ – variable for differential angle (-); ρ – density (kg/m^3); ϵ – turbulent dissipation energy (m^2/s^3); P is power consumption (W); D – Diameter of impeller (m); N – speed of the impeller (rev/min); N_p – Power number

Reference

Alberini, F., Albano, A., Singh, P., Christodoulou, C., Montante, G., Maluta, F. & Paglianti, A. 2024. Fluid Dynamics And Power Consumptions In A Single Use Stirred Tank Adopted In The Pharmaceutical Industry. *Chemical Engineering Research And Design*.

Basavarajappa, M., Draper, T., Toth, P., Ring, T. A. & Miskovic, S. 2015. Numerical And Experimental Investigation Of Single Phase Flow Characteristics In Stirred Tanks Using Rushton Turbine And Flotation Impeller. *Minerals Engineering,* 83**,** 156-167.

Devarajulu, C. & Loganathan, M. 2016. Effect Of Impeller Clearance And Liquid Level On Critical Impeller Speed In An Agitated Vessel Using Different Axial And Radial Impellers. *Journal Of Applied Fluid Mechanics,* 9**,** 2753-2761.

Foukrach, M., Bouzit, M., Ameur, H. & Kamla, Y. 2020. Effect Of Agitator’s Types On The Hydrodynamic Flow In An Agitated Tank. *Chinese Journal Of Mechanical Engineering,* 33**,** 1-18.

Kumaresan, T. & Joshi, J. B. 2006. Effect Of Impeller Design On The Flow Pattern And Mixing In Stirred Tanks. *Chemical Engineering Journal,* 115**,** 173-193.

Rielly, C., Habib, M. & Sherlock, J.-P. 2007. Flow And Mixing Characteristics Of A Retreat Curve Impeller In A Conical-Based Vessel. *Chemical Engineering Research And Design,* 85**,** 953-962.

Tamburini, A., Gagliano, G., Scargiali, F., Micale, G., Brucato, A. & Ciofalo, M. 2019. Cfd Simulation Of Radially Stirred Baffled And Unbaffled Tanks. *Chem. Eng,* 74**,** 1033-1038.