International Journal of Modelling and Simulation

ISSN: 0228-6203 (Print) 1925-7082 (Online) Journal homepage: http://www.tandfonline.com/loi/tjms20

CFD Simulation of Flow Patterns in Dual Impeller Stirred Tank

Thiyam T. Devi & Bimlesh Kumar

To cite this article: Thiyam T. Devi & Bimlesh Kumar (2013) CFD Simulation of Flow Patterns in Dual Impeller Stirred Tank, International Journal of Modelling and Simulation, 33:2, 117-125

To link to this article: http://dx.doi.org/10.2316/Journal.205.2013.2.205-5776

Published online: 15 Jul 2015.

International Journal of Modelling and Simulation, Vol. 33, No. 2, 2013

CFD SIMULATION OF FLOW PATTERNS IN DUAL IMPELLER STIRRED TANK

Thiyam T. Devi* and Bimlesh Kumar*

Abstract

The hydrodynamics of fluid flow inside the stirred tanks are characterized by the flow patterns, which differ with the types of impeller (Rushton, CD-6), number of impellers (more than one) and their clear off distances (S_1, S_2, S_3). Due to the wide application of stirred tanks in process industries, it is required to understand the hydrodynamics of fluid flow for its optimum design. Literature suggests that there exist three most significant flow patterns as parallel, merging and diverging flow in dual impeller–stirred tank at specific impeller clear off distances. In this study, these types of flow patterns have been analysed using computational fluid dynamics (CFD) techniques with multiple reference frame (MRF) impeller model in dual impeller (same diameter) of Rushton and CD-6 impeller since these two impellers act differently in the performance of stirred tanks. From the comparison of Rushton and CD-6 impeller in terms of mean flow and turbulent kinetic energy, it has been observed that there is no distinct difference in the formation of flow patterns, but apparently witnessed strong magnitude in the case of CD-6 impeller.

Key Words

Fluid flow, mixing tank, Rushton impeller, CFD

1. Introduction

It is quite often mentioned that the importance of studying the stirred tank is well known because of its wide applications in several industries like chemical, mineral processing, metallurgical, crystallizers, mixer-settlers, paper and food processing, *etc*. In these industries, the main objective is proper mixing, which is represented by the formation of complex recirculating turbulent flow in the tank due to strong swirling motion of fluid flow. The complexity of fluid flow is greatly increased when more than one impeller in a mixing tank is used and is produced due to the strong interaction between the fluid motions with the stationary walls and baffles inside the stirred tank. Multiple impellers are also commonly used where the tall vessels are required to uniformly distribute the fluid flow motion throughout the stirred tank subsequently reducing the dead zone regions in such tanks. The formation of different fluid dynamic characteristics (flow pattern) is certainly observed in dual Rushton impellers of tall stirred tanks and is different from singular impeller system. And, the efficiency of mixing and fluid characteristics are mainly influenced by many factors like types of impeller, number of impeller, spacing of impellers, *etc*. These different fluid characteristics generated by dual Rushton impellers are stable and also defined as parallel, merging and diverging flow depending upon the off-bottom clearance of lower impeller and spacing between two impellers.

Study of stirred tanks is greatly influenced by using soft techniques due to the spectacular progress of digital applications in research. Computational fluid dynamics (CFD) is also a kind of such techniques widely and successfully adopted by many researchers, scientist, modeller, *etc*. The success story of CFD simulation in the studies of stirred tanks is a long list including the study done by Yapici *et al.* [1] to investigate the effects of type of impeller on flow characteristics with large-eddy simulation (LES) turbulence model and found good agreement with experimental results. Study based on the parameters of CFD simulation was performed by Deglon and Meyer [2] and defined the suitable grid, turbulence model and discretization schemes. Aubin *et al.* [3] also performed CFD simulation based on modelling approach, turbulence model and numerical scheme.

Many studies have been done on same combination of dual impeller systems with Rushton impeller [4]–[10]. Some studies based on different combination of dual impeller systems were performed by various researchers [11]–[13] with Pitched blade impeller and a standard Rushton impeller. And, it has also been verified by many researchers that using impeller more than once gives more enhancing efficiency in the performance of stirred tanks. The multiple impeller system provides better gas utilization, higher inter-facial area and narrower residence time distribution in the flow system compared to a single impeller system [6].

From the knowledge of literature, it has been known that using of non-standard impeller in industries is much lesser than the standard impeller like Rushton impeller whether it may be in single or dual impeller system because of its complex geometrical shape even though it provides better performance in stirred tanks. But it has been

* Department of Civil Engineering, Indian Institute of Technology Guwahati, Guwahati, India; e-mail: {d.thiyam, bimk}@iitg.ernet.in

Recommended by Dr. M. Nakhaeinejad
(DOI: 10.2316/Journal.205.2013.2.205-5776)

understood that the performance of stirred tanks cannot be compromised with the problems faced in designing of such complex geometrical shape since it directly affects the cost and quality of product. The formation of different fluid dynamic characteristics (flow patterns) by dual impeller system is more significant than the single impeller and the application of impeller more than once is very common in several industries. So, it is imperative to introduce a non-standard impeller like CD-6 impeller system (having concave blade shape) of stirred tanks which can be used in several industries for investigating the flow patterns developed by such types of impeller and it is also very mandatory to compare these flow patterns with standard impeller (Rushton impeller) for better understanding the performance of such stirred tanks. Hence, in this study an attempt is being made to characterize the different fluid dynamics developed by same diameter of dual CD-6 impeller system using CFD simulation and these simulated results were being compared with simulated results of Rushton impeller.

2. Numerical Modelling and Geometry

Numerical modelling part in this study defines about the governing equations and governing parameters involved in CFD simulation. The details of these parameters have been analysed in the previous paper of the same authors [14]. In CFD simulation, two main equations are involved as continuity and momentum equation. These equations describe the motion of fluid substances and are solved by iteration process in CFD simulation. In the main momentum equation, one more term called Reynolds stresses (velocity component) by averaging the mean and fluctuating velocity is added, and the new equation is known as Reynolds averaged Navier–Stokes (RANS) equation. To solve this Reynolds stresses, several turbulence models (k-ε; k-ω; Reynolds stress model, RSM; *etc.*) are available based on certain assumptions. k-ε turbulence model is widely adopted model by many researchers and modellers in several fields. In this study, one family of k-ε turbulence model (realizable) is used as it gives satisfactory results with low cost. Realizable k-ε turbulence model is the latest one among the other families (standard and re-normalization group) of k-ε turbulence model considering the swirling fluid motions generated due to the complexity of geometry

Table 1
Values of Constants used in Realizable k-ε Turbulence Model

Constants	C_μ	C_1	C_2	σ_k	σ_ε
Values	0.09	0.43	1.9	1	1.2

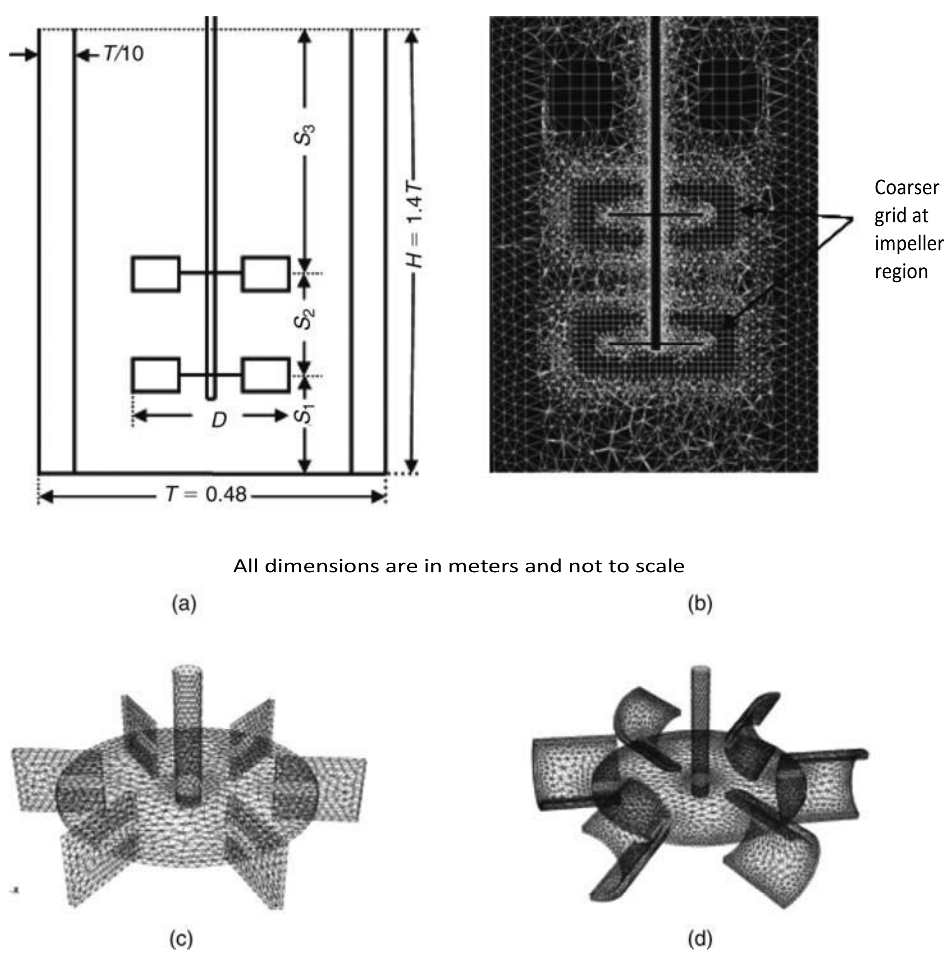

Figure 1. (a) Geometry of stirred tank used in this model, (b) coarser grid at impeller region (iso-surface at $y = 0$), (c) Rushton impeller, and (d) CD-6 impeller.

like near the walls, baffles, *etc.* Constants used to solve the transport equations in Realizable k-ε turbulence model are tabulated in Table 1.

To get significant stable flow patterns, the impeller clear-off distances have been made same with the dimensions taken by Chunmei *et al.* [5]. The axial height of lower impeller from tank bottom is denoted by S_1; spacing between two impellers is by S_2 and distance of upper impeller from top open tank is by S_3. Figure 1 shows (a) geometric dimensions of stirred tanks with notations; (b) the coarser grid at the impeller region (iso-surface at $y = 0$) and (c) and (d) types of impeller (Rushton and CD-6).

Baffled cylindrical tank diameter (T) of 48 cm of dual Rushton impeller and CD-6 impeller with same diameter (D) of 19 cm ($0.4T$) at the speed of 66.6 rpm (N) mounted on a shaft at different axial height was being modelled using CFD Fluent®. The combinations of flow pattern with impeller axial height are given in Table 2. Working fluid is water at 20°C temperature (density = 998.2 kg/m^3; dynamic viscosity = 0.001003 Pa.s) and filled upto 67 cm ($H = 1.4T$). No slip conditions at the boundary of tank wall are considered.

Table 2
Combinations of Flow Patterns with Different Axial Height

Case	Flow Pattern	S_1	S_2	S_3
1	Parallel	$0.40T$	$0.48T$	$0.52T$
2	Merging	$0.40T$	$0.315T$	$0.685T$
3	Diverging	$0.15T$	$0.40T$	$0.85T$

In this study, multiple reference frame (MRF) impeller model is used to model the same combination of dual impeller of CD-6 and Rushton impeller. MRF impeller model of stirred tank defined by CFD simulation in this study is shown in Fig. 2. In this impeller model, the simulation model is divided into two zones as stationary and rotating. In stationary zones, the baffles, tank walls and tank bottom are included, while in the rotating zone, impeller region and impellers itself are involved. In the rotating zone, two separate impeller regions for the two impellers rotate at different frequencies, but the impellers remained at rest. MRF impeller model is designed for complex geometrical dimensions like baffles or other internals of stirred tanks.

The order of discretization refers to the convective terms in the governing equations. The available discretization schemes are central differencing scheme, first- and second-order differencing scheme, Quick differencing scheme, *etc.* [15]. First-order upwind differencing scheme is used in this study and is suitable when the convection dominates and the flow is aligned with the grid. Simulations in this study were typically considered converged when the residuals fell below 1×10^{-4}.

3. Results and Discussion

In this section, the simulated results of mean velocity, turbulent kinetic energy and turbulent dissipation rate for

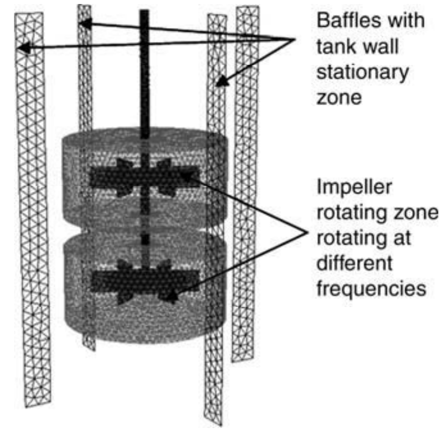

Figure 2. Multiple reference frame (MRF) impeller model of stirred tank.

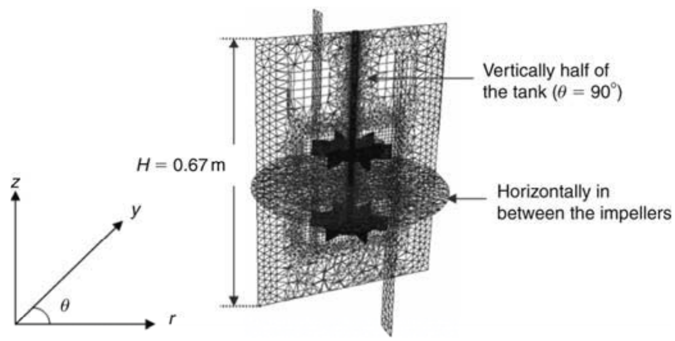

Figure 3. Reference diagram showing the iso-surfaces at the half of the tank ($\theta = 90°$ or $y = 0$) and in between the two impellers for Figures 4 and 6, respectively

Rushton and CD-6 impeller will be presented and discussed in detail. Figure 3 shows the reference grid diagram used for generating various concern and Figs. 4 and 5 of stirred tanks in the following sections.

3.1 Mean Velocities and Vorticity

The performance of a stirred tank is also strongly represented by its mean velocity components which directly defines the mixing efficiency of the stirred tank. In mixing process, the formation of vortex (strong circulating flow) of fluid is needed for its efficiency in mixing, but sometimes it makes worse in the operation as it tends to burst the operation tank if these flow patterns are not stable. So, at this situation, the formation of stable flow patterns is highly recommended. And, generally, in case of Rushton impeller, formation of vortex pair one above and another below the impeller blade is observed for singular impeller. Hence, the point is that how the formation of these vortex pair will be in the case of double impeller and what will be the spacing between this impeller if stable flow patterns are supposed to be achieved. There are three stable flow patterns observed in case of double impeller as parallel, merging and diverging flow according to the formation of

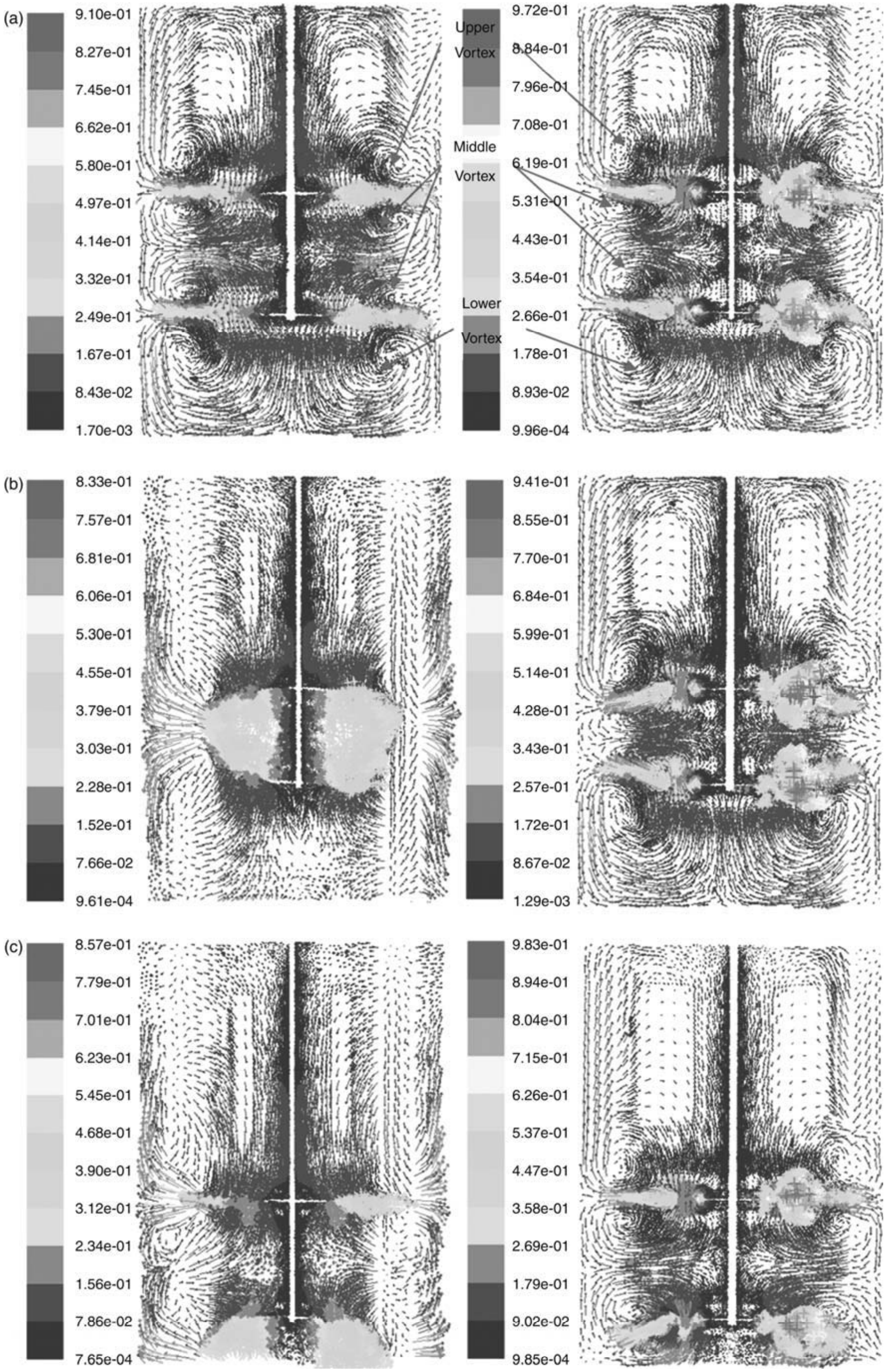

Figure 4. Mean velocity vector plots of Rushton (left) and CD-6 impeller (right) at $\theta = 90°$ along the z-r axis: (a) parallel flow ($S_1 = D = 0.40T,\ S_2 = 0.48T$), (b) merging flow ($S_1 = D = 0.40T,\ S_2 = 0.315T$), and (c) diverging flow ($S_1 = 0.15T,\ D = S_2 = 0.40T$).

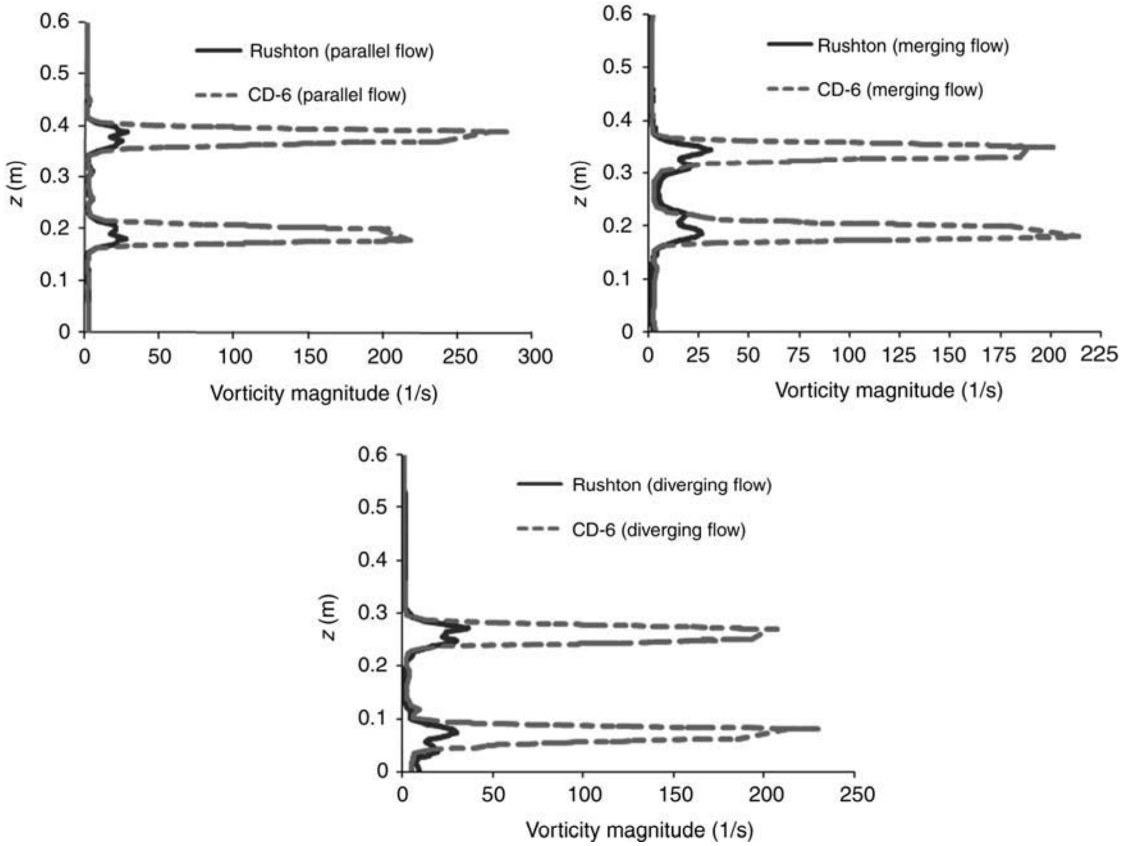

Figure 5. Comparison of vorticity magnitude (1/s) between Rushton and CD-6 impeller for different flow patterns at impeller tip.

flow patterns made by the corresponding spacing between the impellers (Fig. 4).

The vector plots of mean velocity for different flow patterns of Rushton (left) and CD-6 (right) impeller at 90° iso-surface along the z-r axis is shown in Fig. 4 (a–c). The formation of vortex pair in the case of Rushton impeller are observed as one above, one in the middle and another below the impeller blade in the case of parallel flow pattern in Fig. 4(a) (left). These flow patterns are compared with CD-6 impeller and observed similar with the Rushton impeller in Fig. 4(a) (right). In this case, the flow patterns near the impeller region are acted independently in their own flow path. The two impellers made separate stable vortices. And, subsequently, in the merging flow of Rushton impeller, the formation of circular flow pattern (vortex) is seen merges one with another at an elevation approximately midway between the impellers forming two large vortices and in the case of CD-6 impeller, it has been observed little difference with the Rushton impeller forming more clear separation of fluid flow in between the impellers in Fig. 4(b). In case of merging flow of CD-6 impeller, the formation of flow pattern is nearly similar with parallel flow pattern instead of forming merging flow pattern even if the axial height configuration of impellers are kept similar with the configuration of Rushton impeller. In the case of diverging flow, the formation of flow patterns is exactly similar in both cases of Rushton and CD-6 impeller forming three separate vortex rings, one above and another two in between the impeller. The formation of this diverging flow pattern is responsible due to low position of the lower impeller. In this case, the impeller stream follows a path towards and impinges upon the base of the tank. Well-defined two vortexes are made by the above impeller and the lower impeller produces only one vortex ring above the blade and failed to produce below the blade due to the low position of lower impeller. In all the cases of Fig. 4, the magnitude of mean velocity is observed higher in case of CD-6 impeller than Rushton impeller leading to better in mixing. It is also observed that the formation of vortex is higher in number in the case of parallel flow (vortex pair four in number) as compared with other flow patterns (in merging flow, number of vortex pair is two and in diverging, it is three in number) giving more efficiency in mixing of fluid flow inside the stirred tank. The flow patterns observed in all the three cases (parallel, merging and diverging flow) of Fig. 4 were also observed similar with the experimental results of Chunmei *et al.* [5].

The magnitude of flow circulation in turbulent flow regime can be understood by the term "vorticity". It directly relates the rotation of the fluid particles and hence is a factor for efficient mixing. Different types of impeller possess different vorticity magnitude [16]–[19]. Figure 5 shows the comparison of vorticity magnitude between Rushton and CD-6 impeller at impeller tip ($r = 0.095$ m) for different flow patterns. In these figures, it depicts that the magnitude of vorticity is much higher in case of CD-6

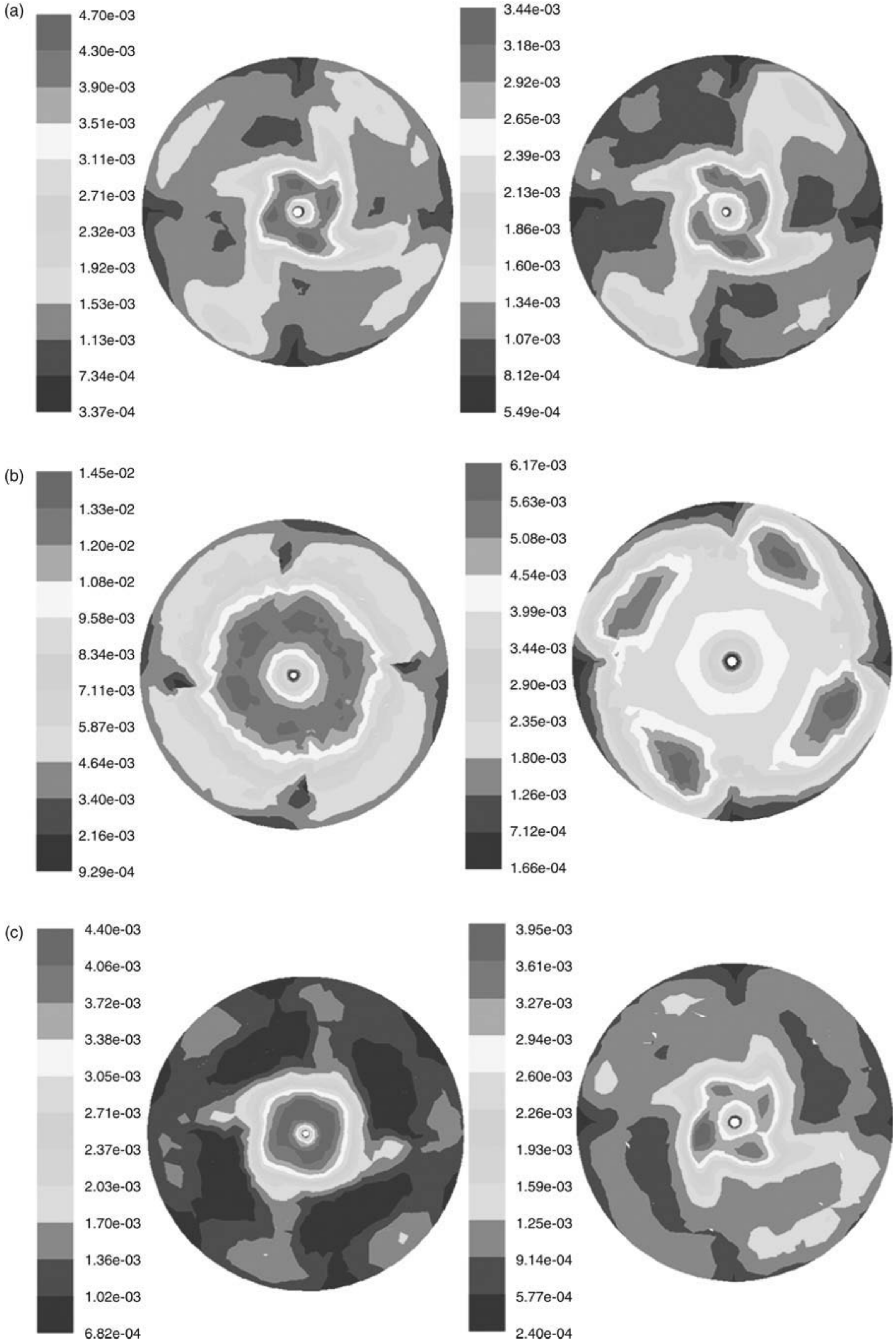

Figure 6. Turbulent kinetic energy contour plots of Rushton (left) and CD-6 impeller (right) in middle of two impellers along the r-θ axis: (a) parallel flow ($S_1 = D = 0.40T$, $S_2 = 0.48T$), (b) merging flow ($S_1 = D = 0.40T$, $S_2 = 0.315T$), and (c) diverging flow ($S_1 = 0.15T$, $D = S_2 = 0.40T$).

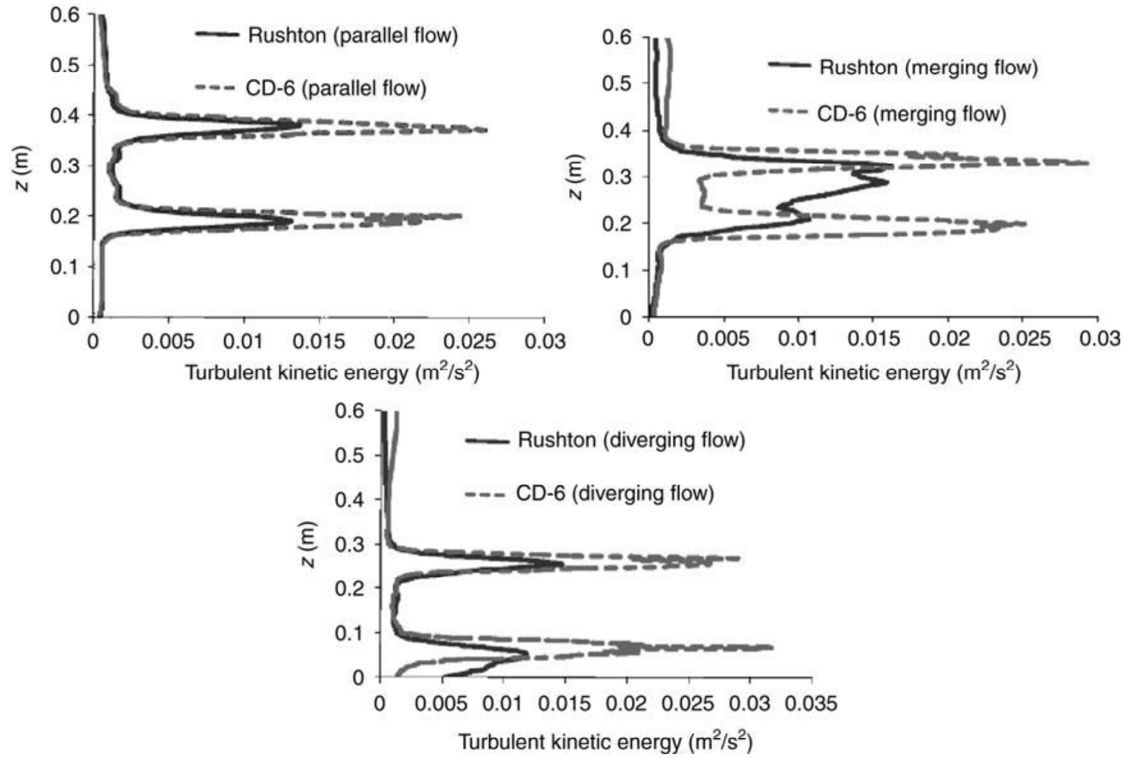

Figure 7. Comparison of turbulent kinetic energy, m^2/s^2 for different flow patterns of Rushton and CD-6 impeller at impeller tip.

impeller (around 90%) than the Rushton impeller in all the three cases of flow patterns. It clearly defines the strong flow circulation tendency by the CD-6 impeller as compared with Rushton impeller.

3.2 Turbulent Kinetic Energy and Dissipation Rate

Turbulent kinetic energy directly defines the performance and efficiency of a stirred tank. The higher magnitude of turbulent kinetic energy represents the strong flow circulation required for proper mixing in mixing operations of process industries.

In Fig. 6(a–c) shows the contour plots of turbulent kinetic energy for different flow pattern of Rushton (left) and CD-6 (right) impeller in the middle of two impellers along the r-θ axis. These diagrams are specially produced to clear understanding of the fluid flow motion in between the impellers when their spacing are differently defined. In parallel flow pattern of Rushton and CD-6 impeller in Fig. 6(a) shows the uniformly distribution of turbulent kinetic energy outside the impeller region, but within the impeller region, the higher accumulation of turbulent kinetic energy is observed. Interestingly, in case of merging flow of Rushton impeller, the turbulent kinetic energy is distributed more uniformly than in case of parallel flow, but near the tank wall its distribution and magnitude are significantly reduced. For CD-6 impeller of merging flow in Fig. 6(b) (right), the uniform distribution and magnitude of turbulent kinetic energy are observed higher near the tank wall than the impeller region and this is a big contrast with Rushton impeller in Fig. 6(b) (left). In the diverging flow pattern of Rushton impeller, the turbulent kinetic energies are uniformly well distributed than in the case of CD-6 impeller in Fig. 6(c). In this case of diverging flow similar with parallel flow, the magnitude of turbulent kinetic energies are observed higher in the impeller region than the other region of stirred tank of Rushton and CD-6 impeller. Figure 7(a–c) shows the comparison of magnitude of turbulent kinetic energy for different flow patterns of Rushton and CD-6 impeller at the impeller tip ($r = 0.095$ m). From these figures, it is observed that the magnitude of turbulent kinetic energy is much higher in the case of CD-6 impeller than the Rushton impeller. This result defines the specific magnitude of turbulent kinetic energy at specific region of stirred tank and from this it has been clearly understood that the higher magnitude of turbulent kinetic energy sticks only near the impeller tip region. And, hence, strong flow circulation is occurred within and near the impeller region as compared with other region of stirred tank. From Figs. 6 and 7, the average magnitude of turbulent kinetic energy is observed higher in case of Rushton impeller than the CD-6 impeller, but if it is specifically observed at particular region (at impeller tip), it has been observed higher in the case of CD-6 impeller than the Rushton impeller.

To sustain turbulent flow, a constant source of energy supply is required, otherwise, turbulence dissipates rapidly as the kinetic energy is converted into internal viscous shear stress. The comparison of turbulent dissipation rate for different three flow patterns of Rushton (left) and CD-6 (right) impeller at impeller tip ($r = 0.095$ m) is shown in Fig. 8. In this figure, in all the three flow patterns, the magnitude of turbulent dissipation rate is observed higher in the case of CD-6 impeller than the Rushton impeller. In

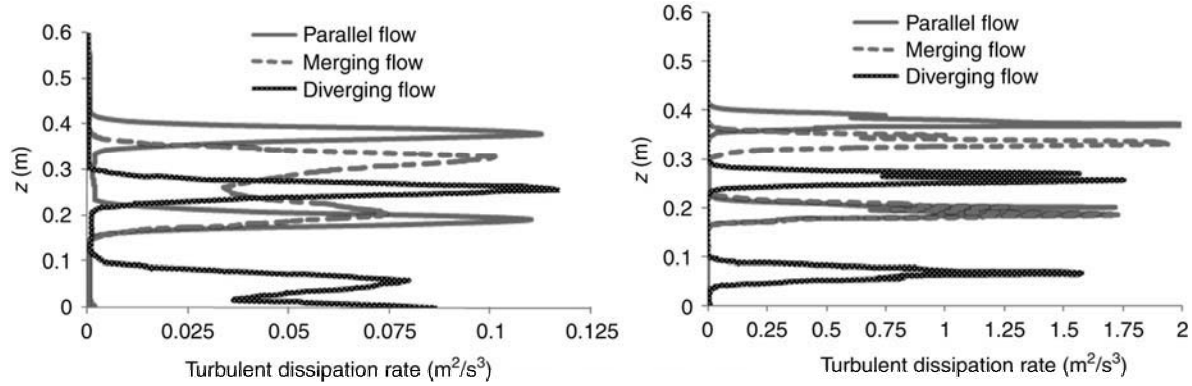

Figure 8. Turbulent dissipation rate, m^2/s^3 for Rushton (left) and CD-6 (right) impeller of three flow patterns at impeller tip.

both cases of Rushton and CD-6 impeller of the comparison of the three flow patterns, it is observed that the magnitude of turbulent dissipation rate is found greater in the case of parallel flow and lowest in the merging flow pattern.

4. Conclusion

The three stable flow patterns (parallel, merging and diverging flow) in both cases of Rushton and CD-6 impeller are observed by numerical modelling of CFD simulation. In the parallel flow pattern of Rushton and CD-6 impeller, four pair of vortex have been observed, one above the upper impeller, two in between the impeller and one below the lower impeller. These vortices are independently produced by the individual impeller indicating independent movement of fluid motion in their own circulating path. In the case of merging flow, the flow pattern of Rushton impeller is observed a little different with the CD-6 impeller needing to make lower the impeller spacing from the case of Rushton impeller for more convincing results. In merging flow, the middle vortex merges one with another of the two impellers leading to no formation of clear vortex in this region of stirred tank reducing in the efficiency of proper mixing. Similar flow pattern is observed in the case of diverging flow of Rushton and CD-6 impeller. The mean velocity, turbulent kinetic energy and turbulent dissipation rate for all the three flow patterns have been analysed and accordingly a comparison is made between the Rushton and the CD-6 impeller. In all the cases of three flow patterns, the distribution of turbulent kinetic energy is accumulated near the impeller region than the other region of stirred tank, but in the case of merging flow of CD-6 impeller, it has been observed that the distribution of turbulent kinetic energy is accumulated near the tank wall than the impeller region making a big contrast with the case of Rushton impeller. An overall conclusion acquired from this study is that the CD-6 impeller produces more powerful flow circulation than the Rushton impeller which leads to better performance of stirred tanks.

References

[1] K. Yapici, B. Karasozen, M. Schafer, and Y. Uludag, Numerical investigation of the effect of the Rushton type turbine design factors on agitated tank flow characteristics, *Chemical Engineering and Processing*, *47*, 2008, 1340–1349.

[2] D.A. Deglon and C.J. Meyer, CFD modeling of stirred tanks: numerical considerations, *Mineral Engineering, 19*, 2006, 1059–1068.

[3] J. Aubin, D.F. Fletcher, and C. Xuereb, Modeling turbulent flow in stirred tanks with CFD: the influence of the modeling approach, turbulence model and numerical scheme, *Experimental Thermal and Fluid Science*, *28*, 2004, 431–445.

[4] A. Bombac and I. Zun, Gas-filled cavity structures and local void fraction distribution in vessel with dual-impellers, *Chemical Engineering Science*, *55*, 2000, 2995–3001.

[5] P. Chunmei, M. Jian, L. Xinhong, and G. Zhengming, Investigation of fluid flow in a dual Rushton impeller stirred tank using particle image velocimetry, *Chinese Journal of Chemical Engineering*, *16*(5), 2008, 693–699.

[6] A.R. Khopkar, G.R. Kasat, A.B. Pandit, and V.V. Ranade, CFD simulation of mixing in tall gas–liquid stirred vessel: Role of local flow patterns, *Chemical Engineering Science*, *61*, 2006, 2921–2929.

[7] L. Xinhong, B. Yuyun, L. Zhipeng, G. Zhengming, and J.M. Smith, Particle image velocimetry study of turbulence characteristics in a vessel agitated by a dual Rushton impeller, *Chinese Journal of Chemical Engineering*, *16*(5), 2008, 700–708.

[8] Y.N. Chiu, J. Naser, K.F. Ngian, and K.C. Pratt, Computation of the flow and reactive mixing in dual-Rushton, *Chemical Engineering and Processing: Process Intensification*, *48*, 2009, 977–987.

[9] R. Zadghaffari, J. Moghaddas, and J. Revstedt, A mixing study in a double- Rushton stirred tank, *Computational and Chemical Engineering*, *33*, 2009, 1240–1246.

[10] M. Taghavi, R. Zadghaffari, J. Moghaddas, and Y. Moghaddas, Experimental and CFD investigation of power consumption in a dual Rushton turbine stirred tank, *Chemical Engineering Research and Design*, *89*, 2011, 280–290.

[11] M. Bouaifi and M. Roustan, Power consumption, mixing time and homogenization energy in dual-impeller agitated gas–liquid reactors, *Chemical Engineering and Processing*, *40*, 2001, 87–95.

[12] M. Jahoda, M. Mostek, A. Kukukova, and V. Machon, CFD modeling of liquid homogenization in stirred tanks with one and two impellers using large eddy simulation, *Transactions of IChemE, Part A, Chemical Engineering Research and Design*, *85*(A5), 2007, 616–625.

[13] S. Woziwodzki and L. Jedrzejczak, Effect of eccentricity on laminar mixing in vessel stirred by double turbine impellers *Chemical Engineering Research and Design*, *88*(11), 2011, 2268–2278.

[14] T.T. Devi and B. Kumar, Analyzing flow hydrodynamics in stirred tank with CD-6 and Rushton Impeller, *International Review of Chemical Engineering*, *3*(4), 2011, 440–448.

[15] E.M. Marshall and A. Bakker, *Computational fluid mixing* (USA: Fluent, Incorporated, 2002).

[16] R. Escudie, D. Bouyer, and A. Line, Experimental analysis of trailing vortices in radially agitated tank, *AIChE Journal*, *50*(1), 2004, 75–86.

[17] A. Khopkar, J. Aubin, C.R. Atoche, C. Xuereb, N.L. Sauze, J. Bertrand, and V.V. Rannade, Flow generated by radial flow

impellers: PIV measurements and CFD Simulations, *International Journal of Chemical Reactor Engineering*, *2*(A18), 2004, 1–17.
[18] A. Delafose, J. Morchain, P. Guiraud, and A. Line, Trailing vortices generated by a Rushton turbine: assessment of URANS and large eddy simulations, *Chemical Engineering Research and Design*, *87*, 2009, 401–411.
[19] Z. Jing, G. Zhengming, and B. Yuyun, Effects of the blade shape on the trailing vortices in liquid flow generated by disc turbines, *Chinese Journal of Chemical Engineering*, *19*(2), 2011, 232–242.

Biographies

Thiyam T. Devi is a research scholar at IIT Guwahati. Her main areas of research are mixing phenomena in stirred tanks and geoinformatics systems.

Bimlesh Kumar got his Ph.D. from the Indian Institute of Science, and is currently working as an assistant professor at the Indian Institute of Technology Guwahati, India. His main research areas are environmental fluid mechanics, sediment transport, and energy planning.