

A Study on Liquid-liquid Mixing in a Stirred Tank with a 6-Blade Rushton Turbine

*R. Zadghaffari¹, J.S. Moghaddas*¹ and J. Revstedt²*

1- Transport Phenomena Research Center, Chemical Engineering Faculty, Sahand University of Technology, Tabriz, Iran.

2- Faculty of Engineering, Lund University Division of Fluid Mechanics, Lund, Sweden.

Abstract

The turbulent flow field generated in a baffled stirred tank was computed by large eddy simulation (LED) and the flow field was developed using the Sliding Mesh (SM) approach. In this CFD study, mixing times and power number have been determined for a vessel agitated by a 6-blade Rushton turbine. The predicted results were compared with the published experimental data. The satisfactory results of comparisons indicate the potential usefulness of this approach as a computational tool for designing stirred reactors.

Keywords: *CFD, Stirred tank, Mixing time, Power, LES, MRF*

Introduction

The mixing or agitation of liquids in stirred tanks being one of the oldest of unit operations, is used by many industries such as chemical, mineral, biotechnological, and food processing for mixing single or multiphase fluids. The optimum design and the efficiency of mixing operations are important parameters on product quality and production costs. The flow motion in stirred tanks is 3-dimensional and complex. In the area surrounding the impeller, the flow is highly turbulent and swirling. A large number of power and mixing time measurements and correlations are available in the literature for impellers of various geometries and for various fluids. These

correlations have been obtained based on laboratory scale measurements, and their scale up to industrial scale mixing devices have always been a matter of concern. In recent years, Computational Fluid Dynamics (CFD) techniques are being increasingly used as a substitute for experiments to obtain the detailed flow field for a given set of fluid, impeller and tank geometries [1-6]. One advantage with CFD based prediction methods is that these do not have scaling up or scaling down problems as these solve the fundamental equations governing fluid flow. So some approximation on the physical phenomena, such as phenomenological models for turbulence, is often required even in CFD simulations. Researchers have

* Corresponding author: E-mail: jafar.moghaddas@sut.ac.ir

employed mainly Reynolds Averaged Navier-Stokes (RANS) techniques to close the equations involved with Reynolds stresses [3,7]. The result of this kind of method is to find good agreement with the experimental measurements in terms of the bulk mean flow in the agitated tank, but they suffer from inaccurate turbulent kinetic energy distribution prediction, especially in the regions close to the impeller due to the isotropic nature of the $k-\varepsilon$ turbulence model [3,8]. Large eddy simulations (LES), first adopted in stirred tank by Eggels[9], have proved to be a good method of investigating unsteady behavior in turbulent flow. Revstedt et al [10] pointed out that LES could provide details of the flow field that cannot be obtained with RANS and corresponding models. Derksen[11,12] used LES with the Smagorinsky subgrid scale (SGC) model to simulate a baffled stirred tank driven by a Rushton impeller.

In CFD, fully predictive simulations of the flow field and mixing time mainly use either the sliding mesh(SM) [13] or the multiple reference frame (MRF) [14] approaches for account impeller revolution. The SM approach is a fully transient approach, where the rotation of the impeller is explicitly taken into account. On the other hand, the MRF approach predicts relative to the baffles [8]. The SM approach is more accurate but it is also much more time consuming than the MRF approach. SM simulation of a stirred tank content homogenization was first published by Jaworski and Dudczak [15], who used the standard $k-\varepsilon$ model and compared the results with the experimental data.

Osman and Varley [16] studied the mixing time in an unbaffled vessel with a Rushton turbine using the MRF approach. The predicted mixing time was found to be up to two times higher than the experimental one and the authors attributed the discrepancies to the under estimation of the mean velocity components near the Rushton turbine.

Jaworski et al [17] studied homogenization in a baffled vessel stirred by a dual Rushton impeller using the MRF approach. Converged solution of the flow field was then used as an input for the solution of the scalar transport equation using the SM approach in order to simulate the time dependent mixing process, but not continuing the computation of the flow field. The predicted mixing time was found to be 2-3 times higher than the measured values, in agreement with [16], they attributed inaccuracies to the under prediction of the mass exchange between the recirculation zones generated by the Rushton turbines and wrongly predicted tangential velocity field [17]. The same authors, Bujalski et al. [18], also predicted these simulations with a denser grid in the regions of high velocity gradients and with a more converged solution. While solving the transient scalar transport equation in a stationary reference frame, improved results were obtained, but the mixing time was still over predicted by about two times. In contrast to these papers, Shekhar and Jayanti [19] successfully simulated flow field and mixing characteristics in an unbaffled vessel stirred by a paddle impeller using low Reynolds $k-\varepsilon$ model for rather low Reynolds numbers.

There are a very low number of CFD based computations of the power consumption and power curve simulation in the literature. S.Jayanti et al [19] simulated the power and mixing time of a Newtonian fluid by a paddle type impeller in an unbaffled vessel by using the SM method and the results are compared with the experimental data.

In most CFD simulations the baffles, impeller disc, and impeller blades are treated as zero thickness walls that are unreal assumptions. Studies have shown that the impeller blade thickness influence the mixing properties [28]. In this work, actual dimensions of a stirred tank reactor were modelled and the thickness of baffles and impeller blades were not neglected. The

mixing process was simulated numerically using LES with a Smagorinsky-Lilly subgrid scale model and flow field, and power consumption and mixing time were simulated in a standard baffled tank reactor stirred with a flat 6-blade Rushton turbine. The results of simulation of power were compared with experimental data [19,20], and the results of simulation mixing time were compared with the empirical correlations [17]. Power number and velocity profile between these two grids were compared with the reported results. The results of the power number and flow field show very good agreement. There are fairly reasonable agreements with the reported values of the predicted mixing time in the literature at similar conditions.

Mathematical Formulation

By combining the conservation of mass equation with the momentum and the energy equations, the necessary flow equations for a mixing system can be developed. In this work we can neglect the temperature increase due to viscous dissipation.

For turbulent flow to obtain the true variation of the velocity field, the conservation equations are solved with the LES Turbulent model.

LES lies somewhere between the DNS (Direct Numerical Simulation) and the RANS (Reynolds Average Navier-Stokes) approaches. Basically, large eddies are resolved directly in LES, while small eddies are modelled. The governing equations employed for LES are obtained by filtering the time-dependent Navier-Stokes equations in either Fourier (wave-number) space or configuration (physical) space. The filtering process effectively ignores the eddies whose scales are smaller than the filter width or computational grid spacing. The resulting equations thus govern the dynamics of large eddies.

A filtered variable is defined by:

$$\bar{\phi}(X) = \int_{\Omega} \phi(X') G(X, X') dX' \quad (1)$$

Where Ω is the fluid domain and G is the filter function that determines the scale of the resolved eddies. The finite-volume discretization provides the filtering operation implicitly:

$$\bar{\phi}(X) = \frac{1}{V} \int_{\Omega} \phi(X') dX', X' \in \Omega \quad (2)$$

Where V is the volume of a computational cell.

The space-filtered equations for the conservation of mass and momentum of an incompressible Newtonian fluid can be written as:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (3)$$

$$\frac{\partial}{\partial t} (\bar{u}_i) + \frac{\partial}{\partial x_i} (\bar{u}_i \bar{u}_j) = -\frac{\partial \bar{P}}{\partial x_i} + \nu \frac{\partial}{\partial x_j} \left(\frac{\partial \bar{u}_i}{\partial x_j} - \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{\partial \bar{\tau}_{ij}}{\partial x_j} + \bar{f}_i \quad (4)$$

$$\bar{\tau}_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j \quad (5)$$

Where \bar{f}_i is the force term and $\bar{\tau}_{ij}$ is the sub-grid scale stress tensor, which reflects the effect of the unresolved scales on the resolved scales.

A common subgrid-scale model is the Smagorinsky-Lilly model [11]. In this method the eddy viscosity is modelled by $\mu_t = \rho L_s^2 |\bar{S}|$. L_s is the mixing length for subgrid scales. In the Fluent program used, L_s is computed using $L_s = \min(Kd, C_s V^{1/3})$. K is the Von Karman constant, d is the distance to the closest wall, V is the volume of the computational cell. In this paper C_s is set to

0.1 [12], and $|\bar{S}| \equiv \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}$, where \bar{S}_{ij} is the rate-of-strain tensor for the resolved scale.

Estimation of Mixing Time

In a real stirred tank, there are large vortices and resulting macro-instabilities, which promote tracer mass exchange through this boundary. The LES model could catch the details of vortices, especially the large vortices, which result in the good prediction of the response curve of the tracer, even down to small Taylor and Kolmogorov micro scale due to sufficiently small mesh size in simulation. For measurement of mixing time, a species injected from the top of the tank and its concentration is simulated at a specific point with a conductivity probe. As shown in Fig. 1 a species is injected just below the free surface, at a horizontal distance of $T/4$ from the vessel wall, opposite the probe (concentration reported point in simulation).

Mixing time is considered as a time taken that variation reported as below $\pm 5\%$ of the fully mixed concentration.

The concentration of species is governed by the following transport equation:

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial x_i}(u_i c) = -\frac{\partial}{\partial x_i} \left(-D_m \frac{\partial c}{\partial x_i} \right) - \frac{\partial}{\partial x_i} (\overline{u_i' c'}) \quad (6)$$

Where c and c' are the mean and fluctuation concentration of the tracer, respectively, and D_m is the molecular diffusion coefficient. A consistent approach is used to represent the turbulent transport process, and hence the turbulent mass fluxes, $\overline{u_i' c'}$, are modelled using the gradient-diffusion approach as:

$$\overline{u_i' c'} = -\Gamma \frac{\partial c}{\partial x_i} \quad (7)$$

where $\Gamma (= \mu_t / \sigma_t)$ is the eddy diffusivity, $\mu_t (= C_\mu \rho k^2 / \varepsilon)$ the turbulent viscosity, σ_t stands for the turbulent Schmidt number and is taken as 0.7, and C_μ is a constant.

Power Consumption

An accurate CFD model should be able to predict important parameters such as the overall power input to a stirred tank. The flow field around the impeller and also the shear stress and the pressure distribution on the impeller blade are resolved after simulation. Then power can directly be estimated from a calculation of the total torque required to rotate the impeller. The torque on each blade can be calculated as:

$$T = \sum_i (\Delta P)_i A_i r_i \quad (8)$$

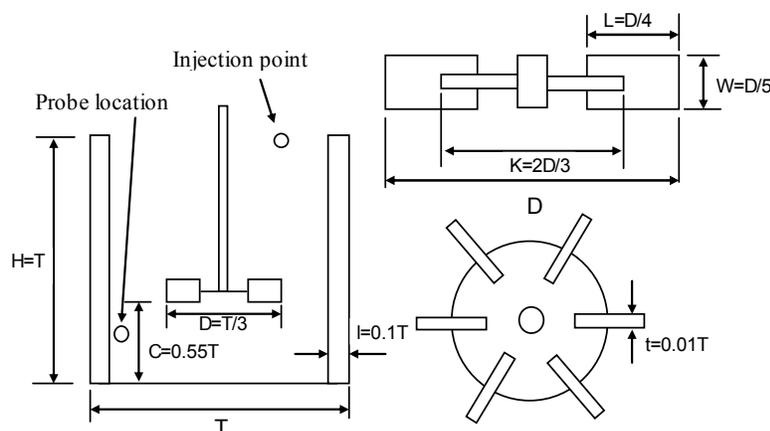


Figure 1. main dimensions of the Rushton impeller and tank used for simulation ($T=30$ Cm)

Where the summation is over the control cells i corresponding to each blade, ΔP is the pressure difference between the front and the back side of the blade at the surface element i , and r_i is the radial distance from the axis of the shaft on which the impeller is mounted. The power required for the rotation of the impeller at a steady rotational speed of N revolution per second for an impeller having m blades is given by:

$$P = 2\pi NmT \quad (9)$$

The power number is then computed as:

$$N_p = \frac{P}{\rho N^3 d^5} \quad (10)$$

Where d is the outer diameter of the impeller[19].

CFD Method

Three-dimensional CFD code fluent, version 6.1, a finite volume based on the fluid dynamic analysis program is used for solving a set of nonlinear equations formed by discretization of the continuity, the tracer mass balance, and momentum equations.

Standard no-slip boundary condition was considered for all solid surfaces. A number of strategies can be used to deal with the movement of the impeller blades as mentioned before. In this study, the MRF solution was used as a starting point. The simulation was then switched to the unsteady SM model [8], with time step set to 0.001 sec, and the second order upwind scheme for discretization and SIMPLE algorithm for pressure-velocity coupling were used.

The system (Fig. 1) consists of a cylindrical standard stirred tank reactor. The grid has 370,997 nodes in the axial, radial and tangential directions. For testing the grid independency, the calculations were repeated with a grid consisting of 612,324 cells and a solution that does not change with a higher number of cells.

Water at 25°C was used as the test fluid ($\rho=10^3 \text{ kgm}^{-3}$, $\mu=10^{-3} \text{ pa s}$). The simulation was run as a transient problem at several impeller rotational speeds in the turbulence regime with an initial condition of zero velocity at all grid nodes.

During the unsteady computations, in order to judge the convergence, the volume integral of the kinetic energy in the tank was calculated. When the pseudo-steady state was reached (after 40s of real time), the computations of the tracer distribution in the vessel were solved together with the flow equations. The power number was then calculated from the pressure distribution on the impeller.

Result and Discussion

The model was able to predict the typical flow patterns observed for a Rushton turbine in a baffled tank as shown in Fig. 2. Along a cross-section through the middle of the tank, the flow field exhibits a characteristic pattern with radial discharge from the impeller. This stream splits into upper and lower circulation zones, with liquid returning axially to the top and bottom of the impeller. A stronger circulation pattern extending over a larger volume of the vessel is created. A low velocity region persists away from the shaft at the top of the vessel. This region decreases with increasing the Reynolds number.

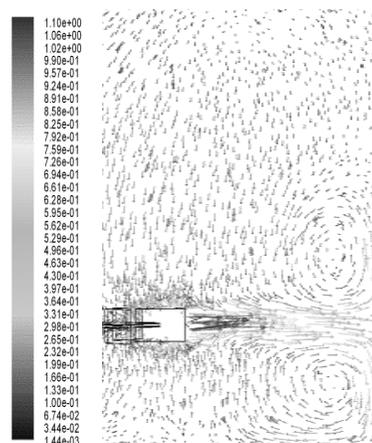


Figure 2. velocity field (m/s) along a cross-section Of the tank, through the middle of the tank In $N_{Re}=250$

A comparison of the predicted time averaged radial, axial and circumferential velocities variations are made with the measured data of Wu and Patterson [26]. Experimental data are taken at different radial distances in a $T=270$ mm vessel and for the impeller rotating at 200 rpm. Geometrical similarity and fully turbulent flow conditions allow experimental and numerical profiles of velocity normalized by the blade tip velocity to be compared directly as shown in Fig. 3 (a,b,c). The results show a good agreement between experimental and simulated data.

Radial profile of the mean radial velocities are displayed in Fig. 3(a). It is observed that impeller stream flows away from the impeller blades, and the velocity varies dramatically in the axial direction. The velocity profiles in the impeller stream become flatter as the fluid moves away from the impeller. This is due to the entrainment of slow moving surrounding fluid into the impeller stream.

Mean circumferential velocity is displayed in Fig.3(b). It is seen that in horizontal sections of the vessel, velocity vectors show counter rotation of the part of the fluid with respect to the impeller. As suggested by Yianneskis et al [27], the presence of the baffles reduces the vessel cross-section, and generates higher values for the circumferential component of velocity and a reduced pressure, which is balanced by the counter flow. Counter flows are stronger where the circumferential component is lower such as far from the impeller, both near to the bottom and the free surface of the vessel.

Mean axial velocity is displayed in Fig.3(c). Results show that in the upper part of the tank fluid moves upward along the wall, and then flow downwards to form a loop in the vertical plane. Another circulation loop exists in the lower part of the tank. The flow toward the impeller blades is stronger in the lower loop than in the upper loop. This is due to the non symmetric boundaries of the gas-liquid interface and the tank bottom. The maximum axial velocity appears in the region where the

fluid is sucked into the impeller blades, and is of the order of magnitude of $0.2 V_{tip}$. As the interface is approached, due to the damping effect of the interface, the velocity magnitude is reduced.

The CFD model predictions do deviate from the experimental data on the lower side of the impeller. Here the CFD results are largely symmetrical, whereas the experimental data is slightly skewed toward the upper side of the impeller. The deviation is due to the lower grid resolution for accounting the computational demand.

Power consumption

The impeller power number (N_p) has been commonly used to check the validity of CFD simulations of the flow in stirred tanks [19].

Fig. 4 shows the pressure distribution in a horizontal plane through the impeller, showing the region of high pressure in front and low pressure behind each blade. With the simulation of pressure difference between both sides of the impeller blade and by using Equations (8-10), the power number(N_p) was calculated and compared with experimental data.

Fig. 5 shows the results obtained for power consumption under turbulent flow conditions that are compared with the experimental data reported by Walas [21].

According to the experimental data N_p is independent from the Reynolds number in the turbulent region and is about 4.8 [21]. Predicted N_p values are in good agreement with the experimental value with a maximum deviation of 3%. The grid resolution and discretization scheme both influence the accuracy of the predicted power number and it is possible to accurately predict the power number using finer grids and high order discretization schemes [29-31].

Mixing Time

The progress of mixing is specific to the flow field which is characterized by a circulation pattern and an effective diffusivity. For

turbulent flows, the molecular diffusivity is augmented by turbulent fluctuations and the effective diffusivity being a strong function of local velocity gradients. There are several empirical relations in the literature that have

been proposed for the prediction the of mixing time. Results of the simulation of mixing time are illustrated in Fig.6 and compared with the empirical correlation [21-25].

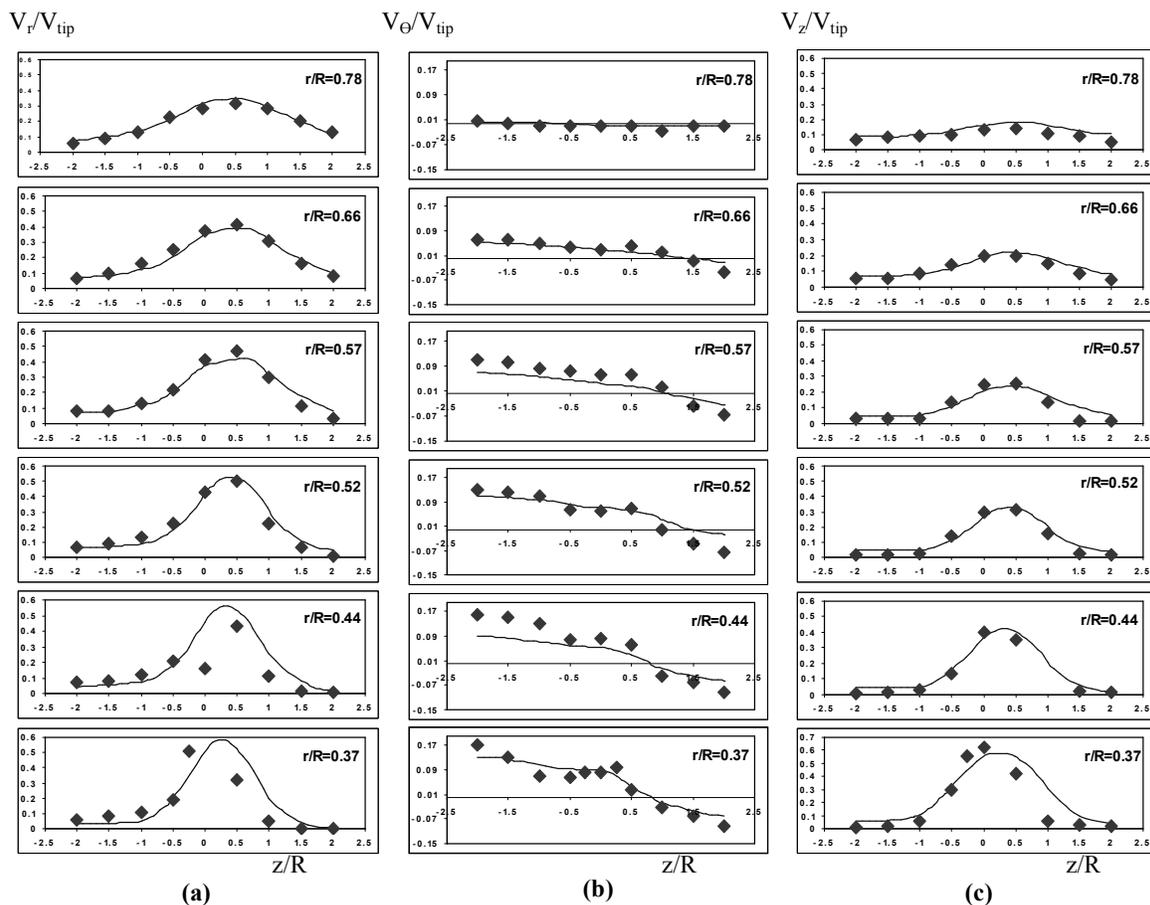


Figure 3. Profile of radial, circumferential and axial component of mean velocity in impeller stream of Rushton turbine: (♦) data by Wu and Patterson [26], and (-) present numerical simulation. (z : axial distance from impeller disk ,R: impeller radius, r: radial distance from impeller shaft)

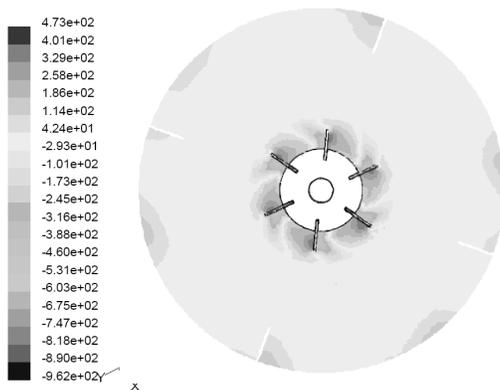


Figure 4. pressure distribution (pascal) in a horizontal plane through the impeller in $N_{Re}=250$

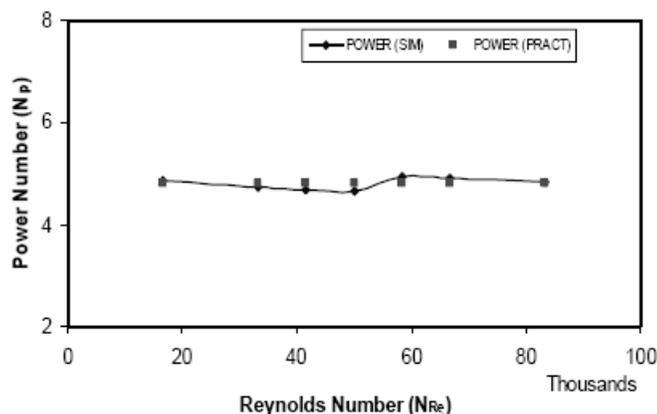


Figure 5. comparison result of the simulated power number (N_p) with experimental data [21].

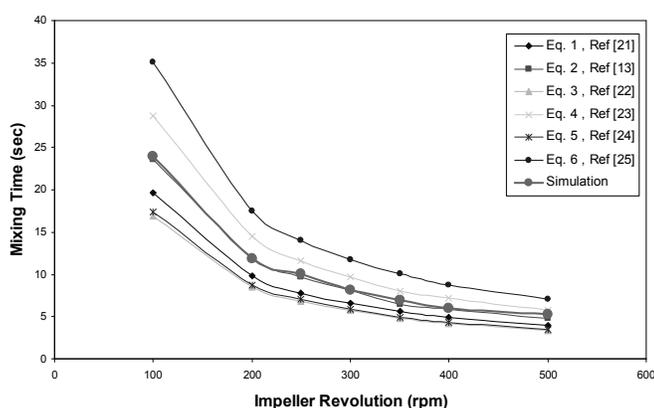


Figure 6. comparison result of simulated mixing time with experimental data

As shown in Fig. 6, by increasing the Reynolds number the stronger radial outflow pushes the species rapidly into the lower and upper recirculation loops and reduced mixing time. They are in fairly reasonable agreement with the values of the calculated mixing time by using Eq. 2, which considers the detailed characteristics of the tank and impeller.

In the present work, the flow field, power consumption and mixing time in a baffled tank stirred by a flat 6-blade Rushton turbine were predicted using the CFD code, fluent 6.1, over a range of impeller Reynolds numbers in turbulent regime by using LES. The flow domain was divided into an inner

rotating and outer stationary domain and by carrying out computations in a time-dependent manner, the effect of the impeller is computed directly. Similarly, actual dimensions of the stirred tank reactor were modelled and the thickness of baffles and impeller blades were not neglected. The tank and impeller geometry had standard dimensions so the simulated data can be compared with experimental data that are available in literature for the flow field [26] and power calculation [21]. Predicted mixing times are compared with empirical correlations [13,21-25]. The computations reported in the present work show reasonable

predictions of the velocity field, power consumption and mixing time in turbulent Reynolds numbers. The accuracy of the predicted variables are influenced by the grid resolution and discretization schemes, and very fine grids and higher order discretization schemes are necessary to limit numerical errors in modelling turbulence.

Nomenclature:

A	Aria, m ²
C	Clearance of impeller, m
d	Distance of computational cell from the closest wall, m
D	Outer diameter of the impeller, m
G	Filter function
H	Liquid height, m
k	Turbulent kinetic energy, m ² /s ²
K ₁	Disc diameter, m
K	The Von Karman constant
l	Baffle width, m
L	Blade length, m
L _s	The mixing length for subgrid scales, m
N	Impeller rotational speed, s ⁻¹
N _p	power number
Δp	Pressure difference, Pa
P	Stirring power input, W/kg
r	Radial direction, m
R	Tank radius, m
R _e	Impeller Reynolds number, $(\frac{N^3 D^5}{\nu})$
t ₁	Blade and baffle thickness, m
t	Time, s
T	Tank diameter, m
u	Mean velocity vector, m/s
V	Volume of a computational cell, m ³
V _{tip}	Impeller tip velocity, m/s
w	Blade height, m
x,y,z	Coordinate vector, m

Greek letters

ρ	Liquid density, kg/m ³
μ	Liquid viscosity, Pa.s
ε	Turbulent dissipation rate, m ² /s ³

Γ	Torque, N.m
Γ _l	Molecular diffusivity, kg/m.s
Γ _t	Eddy diffusivity, kg/m.s
σ _t	Turbulent Schmidt number
μ _t	Turbulent viscosity, Pa.s
τ _{ij}	The sub-grid scale stress tensor, Pa
Ω	The fluid domain
ν	Kinematic viscosity, m ² /s
ν _t	The eddy viscosity, m ² /s

Subscripts

i,j	Coordinate system
t	Turbulent

Superscripts

Overbar	spatial filter
---------	----------------

References

- Dong, L., Johansen, S.T., and Engh, T.A., "Flow induced by an Impeller in an unbaffled Tank: II. Numerical Modelling", *Chemical Engineering Science*, Vol. 49, No. 20, 3511-3518(1994).
- Dong, L., Johansen, S.T., and Engh, T.A., "Flow induced by an Impeller in an unbaffled Tank: I. Experimental", *Chemical Engineering Science*, Vol. 49, No. 4, 549-560 (1994).
- Ng, K., Fentiman, N.J., Lee, K.C. and Yianneskis, M., "Assessment of sliding mesh CFD predictions and LDA measurements of the flow in a tank stirred by a Rushton impeller". *Transactions of the Institution of Chemical Engineers, Chemical Engineering Research and Design*, 76(A):737-747(1998).
- Ranade, V.V., Bourne, J. R., Joshi, J. B., 1991, "Fluid Mechanics and Blending in agitated Tanks", *Chem. Eng. Sci.*, 46, 1883-1893(1991).
- Schmalzriedt, S., Reuss, M., "Application of Computational Fluid Dynamics to simulations of Mixing and Biotechnical Conversion Processes in Stirred Tank Bioreactors", *Proc. of 9th Eur. Conf. on Mixing, Paris*, 171-178(1997).
- Jaworski, Z., Bujalski, W., Otiomo, N., "CFD study of homogenization with dual Rushton turbines-Comparison with experimental results", *Trans IChemE*, 78A, 327-333 (2000).

7. Yeoh, S.L., Papadakis, G., and Yianneskis, M., "Numerical simulation of turbulent flow characteristic in a stirred vessel using the LES and RANS approaches with the sliding/deforming mesh methodology", *Trans IChemE, Part A, Chem Eng Res Des*, 82 834-848, (2004).
8. Mostek, M., Kukukova, A., Jahoda, M. and Machon, V., "Comparison of Different Techniques for Modelling of Flow Field and Homogenization in Stirred Vessels", *Vol. 59, Part 6*, 380-385(2005).
9. Eggels, J. G. M., "Direct and Large-eddy Simulation of Turbulent Fluid Flow Using the Lattice-Boltzmann Scheme", *Int. J. Heat and Fluid Flow*, 17(3), 307-323(1996).
10. Revstedt, J., Fuchs, L. and Tragardh, C., "Large Eddy Simulation of the Turbulent Flow in a Stirred Reactor", *Chemical Engineering Science*, 53(24), 4041-4053 (1998).
11. Derksen, J., Harry, E.A. and Van den Akker, "Large Eddy Simulations on the flow driven by a Rushton Turbine", *AIChE Journal*, 45(2), 209-221(1999).
12. Derksen, J., "Assessment of Large Eddy Simulations for Agitated Flows", *Trans IChemE*, 79(Part A), 824-830(2001).
13. Murthy, J.Y., Mathur, S.R. and Choudhury, D., "CFD Simulation of Flows in Stirred Tank Reactors Using a Sliding Mesh Technique", *Mixing 8, Proceedings of the Eighth European Conference on Mixing*, Institution of Chemical Engineers, Symposium Series No. 136, 341-348(1994).
14. Luo, J.Y., Issa, R.I. and Gosman, D., "Prediction of impeller induced flows in mixing vessels using multiple frames of reference", *IChemE Symp. Ser.*, 136:549-556(1994).
15. Jaworski, Z. and Dudczak, J., "CFD modelling of turbulent macromixing in stirred tanks. Effect of the probe size and number on mixing indices", *Comput. Chem. Eng.*, Volume 22, Supplement 1, 293-298, (24 May 1998).
16. Osman, J. J., Varley, J., "The use of computational fluid dynamics (CFD) to estimate mixing times in a stirred tank", *IChemE Symp Ser*, 146(1999).
17. Jaworski, Z., Bujalski, W., Otomo, N. and Nienow, A.W., "CFD study of homogenisation with dual Rushton turbines-comparison with experimental results. Part I: initial studies", *Transactions of the Institution of Chemical Engineers, Part A-Research and Design*, 78, 327-333(2000).
18. Bujalski, W., Jaworski, Z., Nienow, A. W., "CFD study of homogenisation with dual Rushton turbines-comparison with experimental results, Part II: Using the multiple reference frame", *Transactions of the Institution of Chemical Engineers, Part A - Research and Design*, 80, 97-104(2002).
19. Shekhar, S. M., Jayanti, S., "CFD study of power and mixing time for paddle mixing in unbaffled vessels", *Trans IChem E, Part A*, 80, 482-498(2002).
20. Ludwig, E. E., "Applied process design for chemical and petrochemical", 3rd ed., Gulf Publishing Co., Houston, TX., p. 303(1999).
21. Walas, S.M., "Chemical process equipment", Butterworth - Heinemann Co., USA, p. 292 (1990).
22. Nienow, A.W., "Gas-Liquid mixing studies: A Comparison of Rushton turbines with some modern impellers", *Transactions of the Institution of Chemical Engineers, Part A - Research and Design*, Vol 74, 417-423(1996).
23. Cui, Y. Q., Vanderlans, R. G. J. M., "Compartment Mixing model for Stirred Reactor with Multiple Turbines", *Institution of Chemical Engineers*, Vol. 70, 261-271(1996).
24. Otomo, N., Bujalski, W., Nienow, A.W. and Takahashi, K., "A novel Measurement Technique for mixing Time in an Aerated Stirred Vessel", *Journal of chemical Engineering of Japan*, Vol. 36, No.1, 66-74(2003).
25. Gao, Z., Niu, G., Shi, L. and Smith, J., "Mixing in stirred tank with multiple hydrofoil impellers", *Proceeding of 11th European conference on mixing*, 557-564(2003).
26. Wu, H., Patterson, G. K., "Laser-doppler measurements of turbulent flow parameters in a stirred mixer", *Chemical Engineering Science*, 44, 2207-2221(1989).
27. Yianneskis, M., Popiolek, Z., Whitelaw, J.H., "An experimental study of the steady and unsteady flow characteristics of stirred reactor", *Journal of Fluid Mechanics*, 175, 537-555(1987).

28. Rutherford, K., Mahmoudi, S.M., Lee, K.S., Yianneskis, M., "The influence of Rushton impeller blade and disc thickness on the mixing characteristics of stirred vessels", *Trans. of IChemE* 74, 369-378(1996).
29. Aubin, J., Fletcher, D.F., Xuereb, C., "Modelling turbulent flow in stirred tanks with CFD: the influence of the modelling approach, turbulence model numerical scheme" , *Experimental thermal and fluid science* 28,431-445(2004).
30. Mostek, M., Kukukova, A., Jahoda, M. and Machon, V., Comparison of Different Techniques for Modelling of Flow Field and Homogenization in Stirred Vessels, Vol. 59, Part 6, 380-385(2005).
31. Aoyi, O. and Onyango, M.S., Homogenization energy in a stirred tank. *Chemical Engineering and Processing Journal*, (2007), doi:10.1016/j.cep.2007.10.014