Investigation of Interface Location in Rushton Turbine Stirred Tanks

Arezou. Ghadi, and Azam Sinkakarimi

***Abstract*—**Study the location of interface in a stirred vessel with Rushton impeller by computational fluid dynamic and experimental data was presented. Mean tangential, radial and axial velocities in various points of tank were investigated. Results show sensitivity of system to location of interface and radius of 7 to 10 cm for interface in the vessel with existence characteristics cause to increase the accuracy of simulation.

***Keywords*—** CFD, Interface, Mean Velocity, Rushton impeller.

I. Introduction

MIXING is one of the most common operations in chemical processes and knowledge of fluid flow pattern can considerably help to optimizing the operation. A large number of process applications involve mixing of single phase flow in mechanically stirred vessels. The optimum design and the efficiency of mixing operations are important parameters on product quality and production costs, so being award of the different characteristics such as velocity distribution profiles and turbulence parameters in optimization of using the vessels is important. The flow motion in stirred tanks is 3-dimensional and complex and surrounding the impeller, the flow is highly turbulent. In recent years, computational fluid dynamic techniques increasingly used as a substitute for experiment to obtain the details flow field for a given set of fluid, impeller and tank geometries [1], [2]. In CFD, fully predictive simulations of the flow field and mixing time mainly use either the sliding mesh (SM) [3] or the multiple reference frame (MRF) [4] approaches for account impeller revolution. The MRF approach predicts relative to the baffles [5]. The SM approach is a fully transient approach, where the rotation of the impeller is explicitly taken in to account. 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 [6]. They used the standard k–ε model and compared results with the experimental data.

Rushton turbine is the traditional six blade disc turbine which has been widely used.

Arezou. Ghadi is with faculty member of Ayatollah Amoli Branch, Islamic Azad University, Department of Chemical Engineering (corresponding author to provide phone:+981212517000; fax:+98 1212517043; e-mail: arezoo.ghadi@gmail.com).

Azam. Sinkakarimi, is with Department of Chemical Engineering, Applied Scientific University, Babol, Iran (e-mail: sinkakarimi@yahoo.com).

The flat blade of the Rushton turbine leads to the formation of a pair of high"speed, low"pressure trailing vortices at the rear of each blade [7],[8]. Ruston impeller has been extensively studied as radial pumping impellers in both single phase [9], [10].

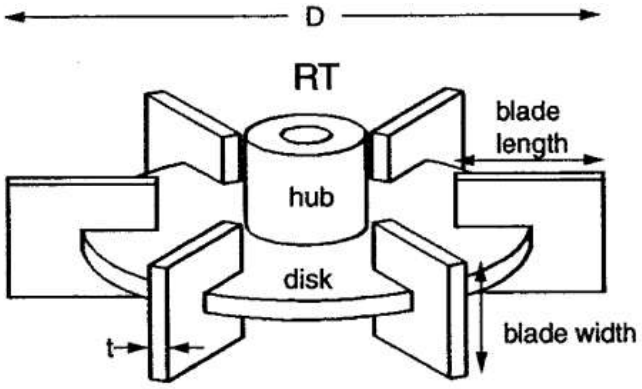

Fig. 1 Rushton impeller

II. Methodology

CFD model is comprised of the numerical solution of conservation equations in laminar and turbulent flow regimes. Hence, theoretical predictions are obtained by the solution of continuity and Reynolds average Navier Stock equations, simultaneously. Continuity and Momentum equations for non compressible Newton fluid are as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_i \frac{\partial u_j}{\partial x_i} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j^2} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

$$\tau_{ij} = \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{3}$$

u_i is the velocity in the ith direction, ρ is the density, p is the pressure and ν is the kinematic viscosity of the fluid and τ_{ij} is stress tensor. For turbulent flow the above set of equations will have to be solved with Direct Numerical Simulation (DNS) to obtain the true variation of the velocity field. The govering equations are time-averaged Navier-Stokes equations and discretize and linearize the results with finite volume method.

A. CFD METHOD

Three-dimensional computational fluid dynamic (CFD) imulations was carried out in order to model the behavior of cylindrical stirred vessels with concave impeller and Rushton

turbine for baffled configurations. Commercial CFD code, fluent version 6.3, was used for solving a set of nonlinear equations formed by discretization of the continuity, the tracer mass balance and momentum equations. A computational grid consisting of two parts: an inner rotating cylindrical volume inclosing the turbine, and an outer, stationary volume containing the rest of the tank. The structured grids, composed of non-uniformly distributed hexahedral cells, were used in the two parts. The grid used in the impeller region was dandified to get a more accurate description of the impeller. The total grid nodes numbers are 500000 in the tank. In this study, first order upwind scheme for discretization also the SIMPLE algorithm for pressure velocity coupling was used. Water at 25°C was used as the test fluid ($\mu=10^{-3}$Pa.s, $\rho=998.2$ kgm^{-3}). Dimensions of the stirred tank and details of Rushton impeller are shown in table I. Fig. 2 shows inner volume of Rushton impeller with exahedral mesh for clockwise rotating.

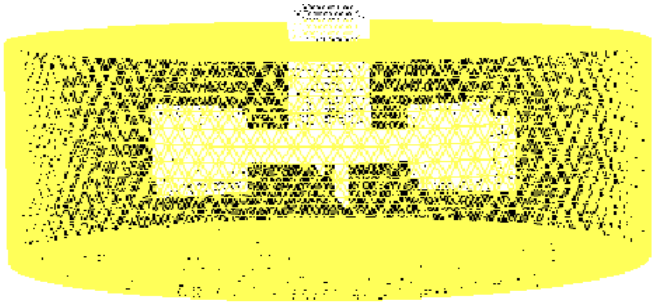

Fig. 2 Inner volume of Rushton impeller with hexahedral mesh forclockwise rotating

TABLE I
DIMENSIONS OF STIRRED TANK AND RUSHTON BLADE

Impeller	symbol	Dimension
Tank diameter(m)	T	0.3
Impeller diameter(m)	D	0.1
Disk diameter(m)	D_0	0.66
Disk thickness (m)	x	0.0035
Blade width (m)	w	0.25
Blade length (m)	l	0.25
Blade thickness (m)	t	0.002
Blade angle(degree)		45
Bottem clearance(m)	C	0.01

No vertical lines in table. Statements that serve as captions for the entire For investigate the location of interface in Rushton impeller, cylindrical has been used with 6.5 cm height and variable radius which surrounded symmetrical the blades. Values of cylinder radius have been adjusted on 5.75, 7.5, 9, 10.5 and 11.75. These radiuses cover respectively the closest distance to the impeller till closest distance to the baffle.

III. RESULTS AND DISCUSSION

In Figs. 3 to 14 simulation results for comparing with experimental data have been presented with laboratory work of Wu and Patterson [11] in r/R of 0.38, 0.5, 0.6 and 0.7 that R is tank radius and r is radial distance from the blade. Radial, angular and axial velocity profiles have been provided in the form of normalized by tip speed of impeller in terms of 2Z/W that Z is axial distance from impeller disk and W is blade height of Rushton impeller. The impeller rotation speed is considered 400rpm. Time step is 0.001sec and to control of reaching quasi steady state drawing the figures of kinetic energy integral is used. Radial velocity variations at different distances of interface have been presented in Figs. 3 to 6.

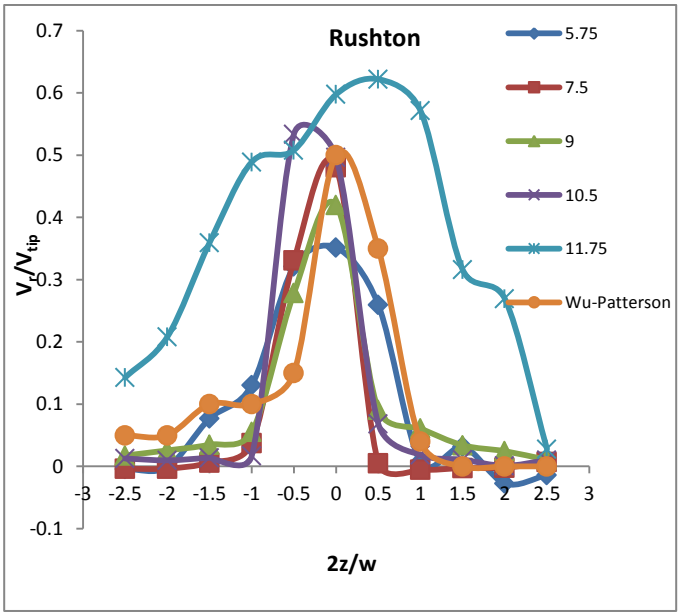

Fig. 3 Normalized mean radial velocity for different distances of interface in r/R=0.38

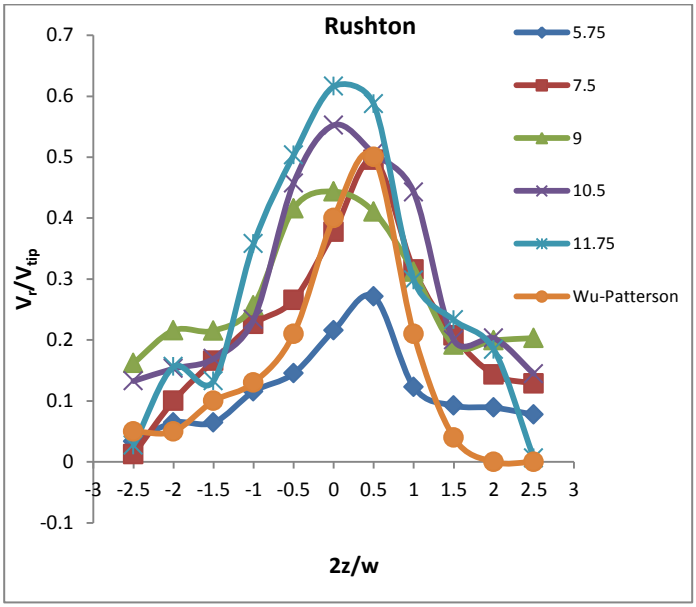

Fig. 4 Normalized mean radial velocity for different distances of interface in r/R=0.5

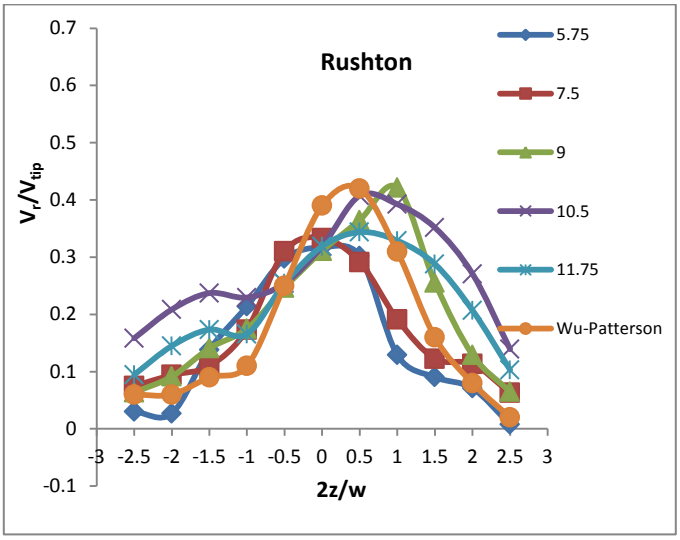

Fig. 5 Normalized mean radial velocity for different distances of interface in r/R=0.6

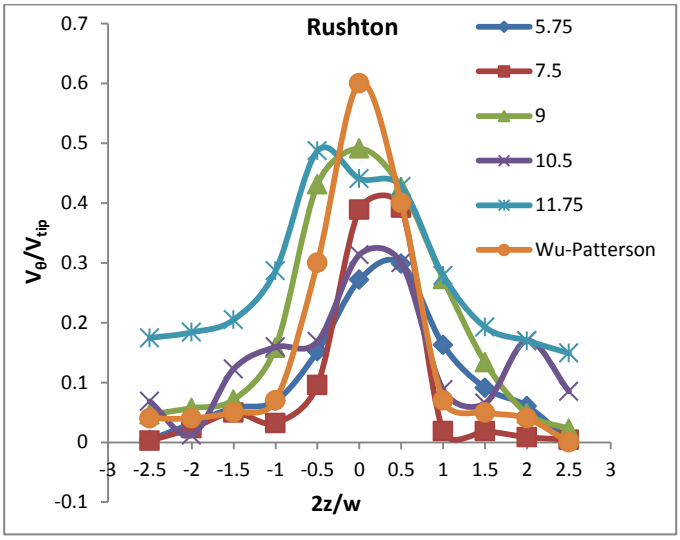

Fig. 7 Normalized mean tangential velocity for different distances of interface in r/R=0.38

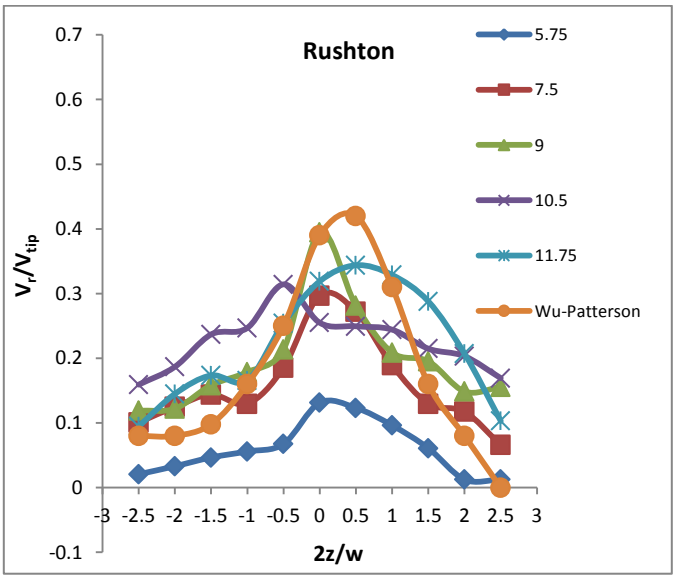

Fig. 6 Normalized mean radial velocity for different distances of interface in r/R=0.7

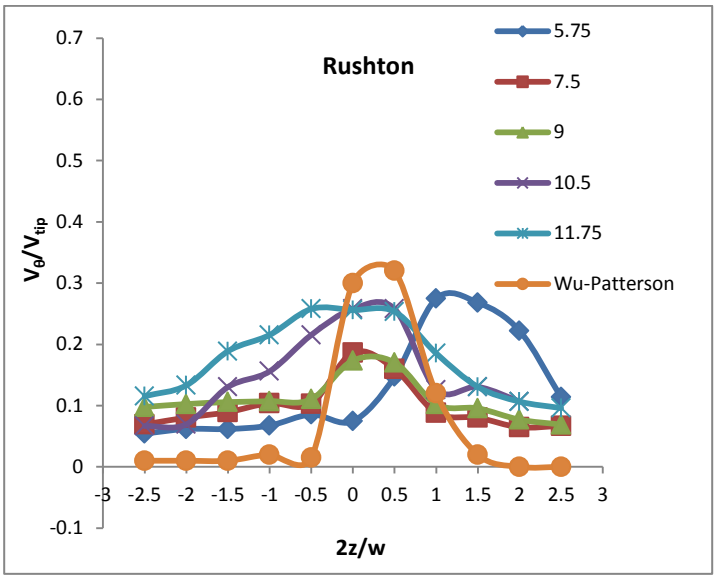

Fig. 8 Normalized mean tangential velocity for different distances of interface in r/R=0.5

According to the figs can be seen the curves obtained using different interfaces have been able to predict the radial velocity curves. With radial movement from impeller toward the tank wall speed decreases gradually and figs are flatter. In case of locating interface in the farthest and nearest distance from the impeller (radiuses 5.75 and 11.75 cm) simulation results comparing to experimental results have higher values of errors but other cases, to determination of radial velocity have been shown less than errors among which 5.75 and 9cm radiuses are more consistent with experimental results. In Figs. 7 to 10 axial variations of tangential velocity are presented. All states except near the blade and close to the tank wall have been able to predict the shape of the tangential velocity. The results of the interfaces between the radius of 7.5 to 9 cm show better performance. Radius 10.5cm after the second and third models has less error.

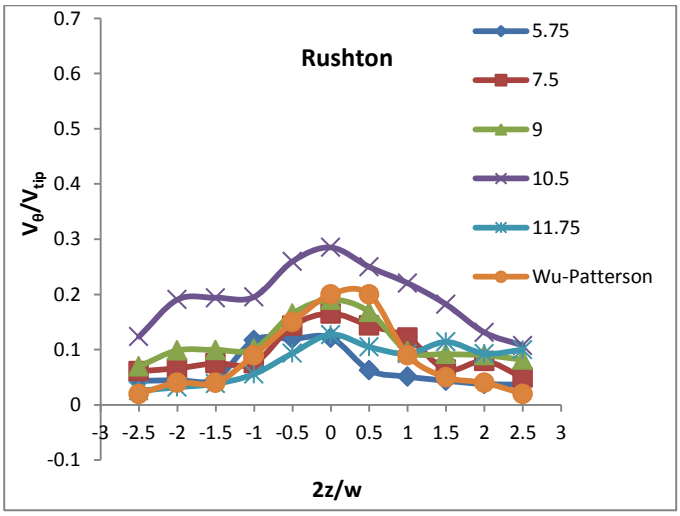

Fig. 9 Normalized mean tangential velocity for different distances of interface in r/R=0.6

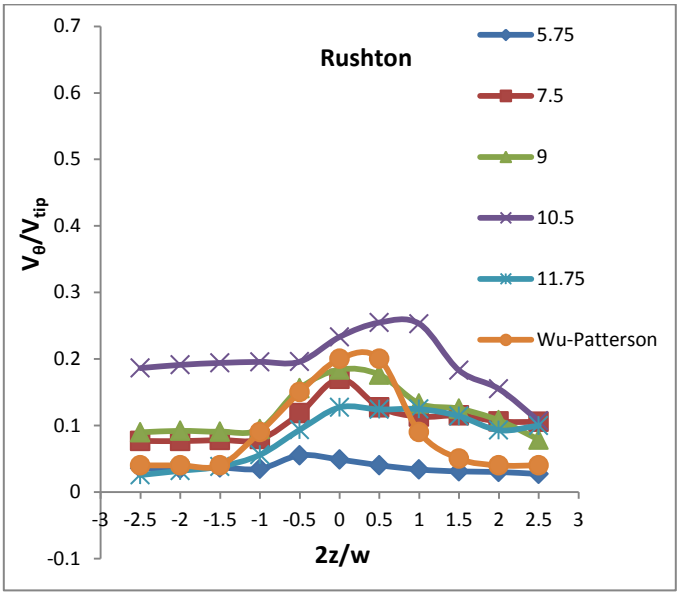

Fig. 10 Normalized mean tangential velocity for different distances of interface in r/R=0.7

Figs. 11 to 14 are shown the axial velocity in different radiuses. Maximum axial velocity is observed in the area that fluid drawn to inside impeller flow. According to thefigures, it is seen that interfaces in 5.75 and 11.75 cm have maximum errors in axial velocity distribution. Also interface in radius of 10.5cm although has less error than the first and fifth models, but has more errors than the second and third models. So the second and third models have the best performances that with increasing ratio of r/R are almost show the same errors.

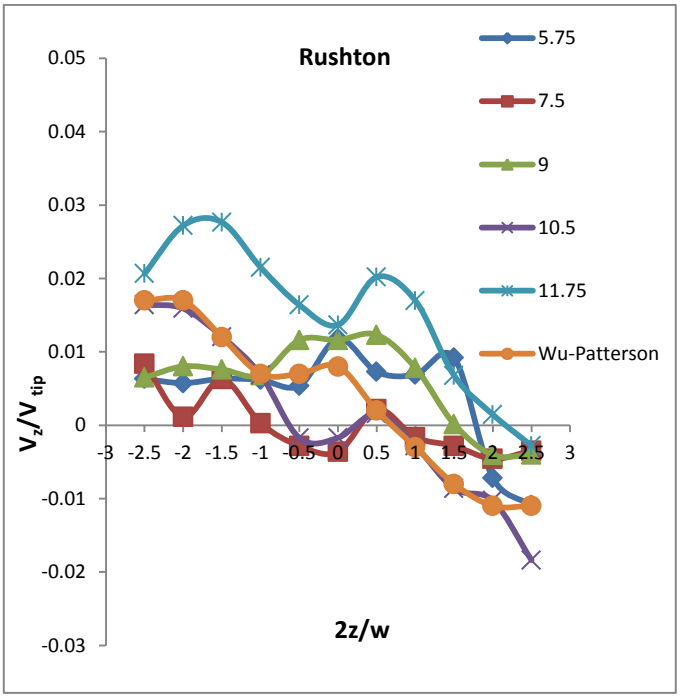

Fig. 11 Normalized mean axial velocity for different distances of interface in r/R=0.38

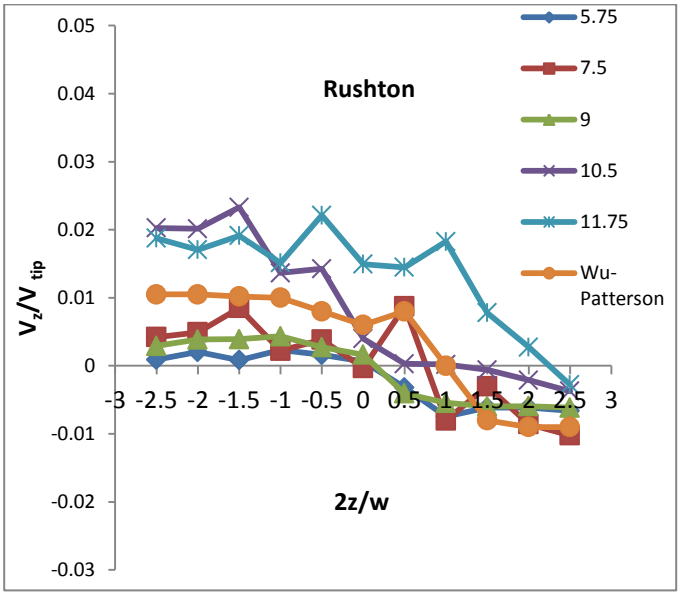

Fig. 12 Normalized mean axial velocity for different distances of interface in r/R=0.5

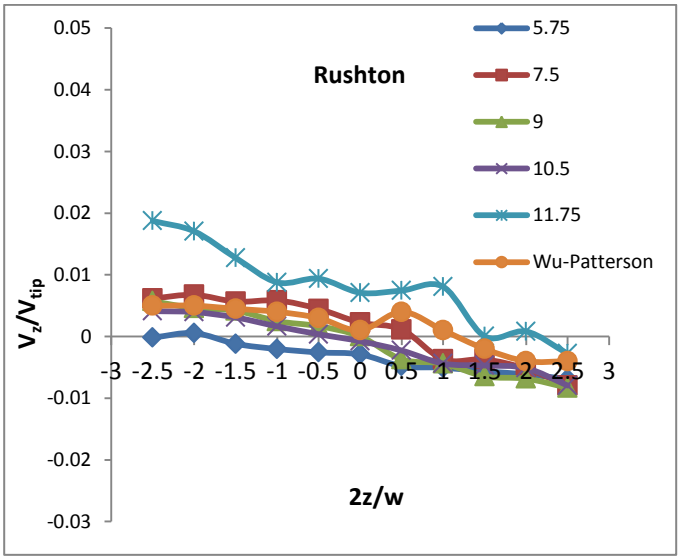

Fig. 14 Normalized mean axial velocity for different distances of interface in r/R=0.7

IV. Conclusion

Location of interface in baffled stirred vessel with Rushton impeller was studied and results compared with experimental data. Senility of the subject to get results with the highest accuracy and the least error cause to investigating the axial, radial and tangential velocities, turbulent energy dissipation rate in different points of the tank. As it shown most of the cases an predict the pattern of Rushton but for nearest and farthest distances from the impeller results are very weak. So according to the simulation and experimental results, the best location of interface for Rushton impeller in a tank with 0.3 m diameter is cylindrical with radius of 7 to 10 cm.

REFERENCES

[1] Ranade VV, Bourne JR, Joshi JB., 1991, Fluid mechanics and blending in agitated tanks. Chem Eng Sci (46), 1883–1893.

[2] Ng,K., Fentiman, N.J., Lee,K.C. and Yianneskis, M., 1998, Assessment of sliding mesh CFD predictions and LDA measurements of the flow in a tank stirred by a Rushton impeller, Trans IChemE, Part A, Chem Eng Res Des, 76(A8), 737–747.

[3] Murthy JY, Mathur SR., 1994, CFD simulation of flows in stirred tank reactors using a sliding mesh technique. IChemE Symp Ser ;136:341–8.

[4] Luo JY, Issa RI, Gosman D., 1994, Prediction of impeller induced flows in mixing vessels using multiple frames of reference. IChemE Symp Ser ;136:549–56.

[5] Mostek M, Kukukova A, Jahoda M, Machon V., 2005, Comparison of different[techniques for modelling of flow field and homogenization in stirred vessels. Chem Pap;59 (Part 6):380–5.

[6] Jaworski Z, Dudczak J., 1998, CFD modelling of turbulent macromixing in stirred tanks. Effect of the probe size and number on mixing indices. Comput Chem Eng; 22 (Suppl.1): 293–8.

[7] Van't Riet, K., Smith, J.M., 1973, The behaviours of gas-liquid mixtures near Rushton turbine blades, Chem. Eng.Sci. (28), 1031-1037.

[8] Van't Riet, K., Smith, J.M., 1975, The trailing vortex system produced by Rushton turbine agitators, Chem. Eng.Sci. (30), 1093-1105.

[9] J.Y. Oldshue, 1983, Fluid Mixing Technology, McGraw Hill, New York

[10] K.C. Lee, M Yianneskis, 1998, Turbulence properties of the impeller stream of a Rushton turbine, Am. Inst. Chem. Eng. J. 44, 13-24.

[11] Wu,H., Patterson,G.K., 1989, Laser"Doppler Measurements o Turblent-Flow Parameters in a Stirrred Mixer, Chemical Engineering Science.,44, (10).2207-2221.