www.arpnjournals.com

STUDY OF FLUID FLOW AROUND IMPELLER BLADES IN RUSHTON TURBINE IN A BAFFLED VESSEL USING COMPUTATIONAL FLUID DYNAMICS

R. Thundil Karuppa Raj, Aman Deep Singh, Shreyak Tare and Satwik Varma
School of Mechanical and Building Sciences, VIT University, Vellore, India
E-Mail: thundil.rajagopal@vit.ac.in

ABSTRACT

The present study summarizes the design of the Rushton turbine and simulates the three-dimensional turbulent flow around the impeller blades of Rushton turbine in a stirred baffled vessel. The 3-dimensional flow around the impeller interacts with the baffles as the impeller rotates. This generates extremely complex flow physics within the vessel being stirred. The simulations are carried out using the Multiple Reference Frame (MRF) approach. The 3-dimensional model of the Rushton turbine and vessel is modelled as per the literature of Ranade *et al.* using SolidWorks package. The Rushton turbine with impellers is discretized into smaller finite volume domains using ICEM CFD pre-processing tool. Hexahedral grids are generated through the turbine and the impellers. The hexahedral cells generated are used to capture the boundary layer phenomenon more accurately compared to the tetrahedral cells for the same cell count. The flow physics involves both stationary and rotating domains and proper interface is provided between these domains so that the flux transfer is more uniform. A finite volume CFD code ANSYS CFX 12.1 is used to capture the 3-dimensional flow fields by solving the continuity, momentum and energy equations using RANS algorithm. A second order upwind scheme is employed for better accuracy. Shear Stress Transport (SST) k- ω turbulence model is used to capture the fluctuating quantities generated due to the turbulence created. SST k- ω has a blending function which ranges from 0 to 1 and can act both s as Standard k-ω and Standard k-ε turbulence models. The flow field near the walls is captured using Standard k-ω models and the flow physics at the free stream region is captured using Standard k-ε model. The numerically predicted results are compared with the experimental data available in the literature. Upon validation, the baffled vessel is varied for different rotational speeds and numerically analysis is carried out. The radial velocity contours and streamlines along with the turbulent kinetic energy are presented at the mid plane of the turbine rotational speeds. The effect of baffles is studied with respect to the flow optimization thereby increasing the reliability and life time of the Rushton turbine.

Keywords: impeller, rushton turbine, baffles, CFD, turbulent flow.

Nomenclature

C	Impeller clearance (m)
D	Impeller diameter (m)
H	Height of the liquid (m)
T	Tank diameter (m)
LES	Large Eddy Simulation
LDV	Laser Doppler Velocimetry
PEPT	Positron Emissive Particle Tracking
Ur	Radial velocity (ms^{-1})
Utip	Impeller tip velocity (ms^{-1})
W	Width of the baffle (m)
Ө	Angle from the blade (°)
μ	Viscosity ($kg\,m^{-1}s^{-1}$)
ρ	Density ($kg\,m^{-3}$)

INTRODUCTION

Mixing of liquids in tanks or vessels is one of the processes that have very diverse applications in the industrial fields. It is carried out in various industries some being the chemical, pharmaceutical, food processing units, bio-technological processes, etc. This mixing may be of different types and the fluids to be mixed may vary accordingly but the reasons behind the mixing remain the same being effective mass and heat transfer, homogenizing the fluid properties throughout the mixture, to prevent the settling of the particles at the bottom, emulsifying and so on. Therefore for carrying out these processes effectively the hydrodynamics of the fluid and level of mixing needs to be studied in detail. Most common stirring technique is the use of impellers in the tanks for stirring purposes. These impellers may be of different types such as a Rushton turbine impeller, Pitched blade turbine etc. Baffles are used in the impeller stirred tanks so as to encourage efficient mixing by the process of dispersion and diffusion.

Different flow measurement techniques such as Laser Doppler Anemometer (LDA) [1], Laser Sheet Illumination (LSI), Positron Emission Particle Tracking (PEPT) [2] and Particle Image Velocimetry [3] have been used in the past to study the highly turbulent, three-dimensional flow structures in impeller-stirred vessels. These experiments are however time consuming as well as economically infeasible. The purpose of the mixing operation in the stirred tank governs the decisions on the configuration of the system. The main design parameters of the impeller-tank configuration are the diameter of the impeller, impeller tip clearance, number of baffles, tank diameter, number of impellers used, etc. Different fluids require different types of mixing which depends on the geometrical factors of the impeller used. Proper choice of material as well as geometry leads to efficient mixing process as well as cost effectiveness of the process. The developments in the field of Computational Fluid

www.arpnjournals.com

Dynamics (CFD) have resulted in an increasing interest in the numerical simulation of the flow in impeller-stirred vessels. This simulation is a complicated study because of the absence of a single frame of reference for computation. This makes the CFD simulation of the tank a challenging process but still an effective and cost efficient when compared with the experimental techniques.

Various techniques have been used in the past by researchers to model the flow near impeller blades in vessels that are stirred. The Multiple Frames of Reference method was introduced by Luo *et al*., [4]. In this the tank is divided into a rotating and a stationary frame. The rotating reference frame comprises of the impeller and the flow around it, whereas the stationary frame includes the tank, baffles and flow outside the impeller frame. Constant impeller position with respect to the baffles is used for meshing purpose. This leads to the simulation of quasi-steady flow field in tank. Bartels *et al*., [5] used this technique to simulate turbulent flows in stirred tank employing Rushton turbine impeller by utilizing DNS and RANS.

An approach called clicking mesh method is used by K. Yapici *et al*., [6]. In this research, the effect of impeller clearance and disk thickness of the impeller on the power number has been studied. Another explicit model is the Momentum Source Method [7] based on aerofoil aerodynamics. This method, however, is useful in the case where axial flow dominates the field as in opposition to the radial flow dominated cases of a Rushton turbine with small clearances.

Montante *et al*., [8] simulated the effect of varying clearances on the power number and circulation patterns for the Rushton turbine impeller using the k-ε turbulence model. Large eddy simulation of mixing in a baffled tank was presented by R. Zadghaffari *et al*., [9]. The impeller rotation was modelled using sliding mesh technique. The results of the CFD analysis were evaluated in terms of the predicted flow field, mean velocity components, power number, mixing time, turbulent kinetic energy etc using published data. Subsequently, the effects of varying injection positions on the flow results have been investigated. H. Singh *et al*., [10] present a study of various turbulence models namely k–ε, SST, SSG-RSM and the SAS-SST models, for predicting turbulent flow in a Rushton turbine stirred tank.

A. Ochieng *et al*., [11] analysed the flow pattern of the fluid at various clearance lengths and also calculated mixing time for each set-up. They also looked for the effect of draft tube on liquid surface pattern to obtain the most favourable clearance at a height of 0.15T and most favourable flow field pattern for which mixing time was minimized. Taghavi *et al*. [12] experimentally studied mixing power consumption and flow regimes in dual Rushton impeller stirred tank in both single and two phase condition and then performed its CFD analysis. Power consumption of lower impeller blade was found to be more than that of upper impeller blade. H.S. Yoon *et al*. [13] performed the scaling of flow with increasing Reynolds number in Rushton turbine stirred impeller.

They computed the changing of radial, axial and circumferential velocities with changing Reynolds number. Also, the changes in various vortex parameters with Reynolds number were studied. M. Campolo *et al*. [14] investigated the role of large-scale convective structures in promoting mixing in stirred tank vessels with four baffles and a six bladed Rushton turbine impeller. The Eulerian-Lagrangian approach was used to numerically investigate dispersion of fluid particles. The three-dimensional, time-dependent, fully developed flow field was calculated using a RANS solver with k–ε turbulence model.

Schafler *et al*., [15] discusses the flow field in the Rushton turbine of stirred tank reactors using numerical simulations. The study used full refractive index matching and laser Doppler Velocimetry for modelling reaction processes for a reliable design of equipment. Schafler produced the angle resolved mean flow, velocity and energy dissipation profiles.

Ranade *et al*., [16] uses the approach of computational snapshot for quantifying the flow produced by the six bladed Rushton turbines and 4 bladed pitched bladed turbines separately in a turbulence field. It aimed to study the trailing vortices behind the blades and CFD model applicability for simulation flow in complex stirred reactors using SIMPLE Algorithm. Ranade and Dommeti *et al*. proposed the approach of snapshot and Ranade *et al*. (1997) discusses about the merits and demerits of different approaches like CSA and sliding mesh for capturing the flow.

METHODOLOGY

In this study a six bladed Rushton Turbine is placed in a cylindrical tank with four baffles along circumference of the tank.

Geometric modelling

The Rushton impeller model with fluid domain is modelled using SOLIDWORKS, Computer Aided Designing tool based on the parameters given. The vessel diameter kept as T=150 mm and the related dimensions relations are mentioned as in Table-1. The thickness of the blade and the disc was kept as 2 mm as shown in Figure-1 below:

Table-1. Impeller dimensions.

Aspect	Relation
Impeller diameter	D=T/3
Blade width	D/5
Disc diameter	D/4
Impeller clearance	T/3

Figure-1. Model of the Rushton Impeller.

The impeller was positioned in a cylindrical domain enclosing the turbine along the length. In order to introduce the dispersion and diffusion mixing in the system, four baffles at 90 degree inclination are incorporated while modelling the domain vessel as shown in Figure-2.
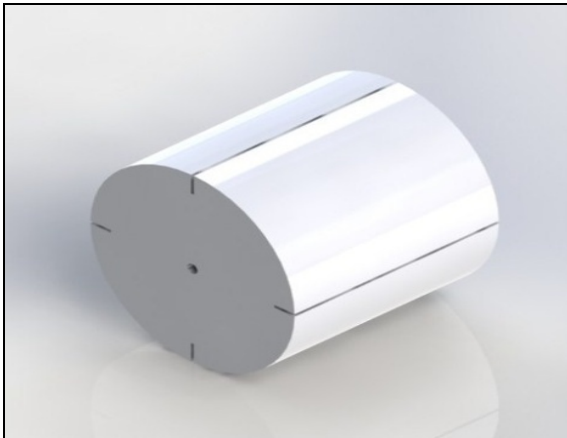

Figure-2. Model of the cylindrical vessel with baffles.

The relations for the cylindrical vessel modelling are mentioned as follows in Table-2.

Table-2. Casing dimensions.

Aspect	Relation
Diameter	T
Height	T
Baffle Width	T/10

GRID GENERATION

The Rushton turbine with impellers is discretized into smaller finite volume domains using ICEM CFD pre-processing tool. Hexahedral grids are generated through the turbine and the impellers, as shown in the Figure-3(a) to keep the cell count feasible for analysis keeping the

account of numerical accuracy. In order to get a detailed flow in the region surrounding the impeller, an enclosure is made which has fine mesh and represents the hub. The technique of MRF is used to capture the flow in the study. In this technique, the hub is kept as stationary but the fluid around the hub in the domain is set to be rotating domain. The hub is also defined as rotationary wall as that of the fluid domain. The fluid domain is resolved into Fluid_Rotor and Fluid_Stator domains. The interface between the rotor and stator domains in the fluid region is defined with the help of Frozen_Rotor Interface with no pitch change. From the literatures, it is found that prism layers would be created near the stator-rotor interface with tetrahedral meshes for uniform flux transfer across the domains. Hence in this study in order to have better uniform flux transfer hexahedral mesh is created over the Fluid_Rotor and Fluid_Stator domains as well across the stator_rotor interface. The volumetric mesh with interfaces, baffles, hub, blades and fluid domains are shown in Figures 3(b) and 3(c) for better visualization.
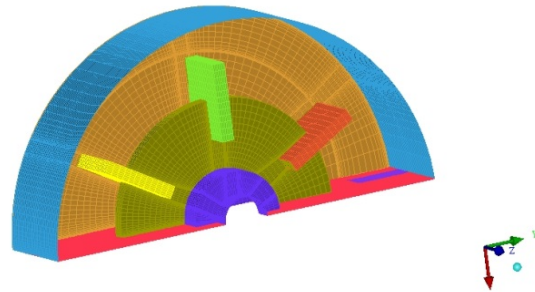

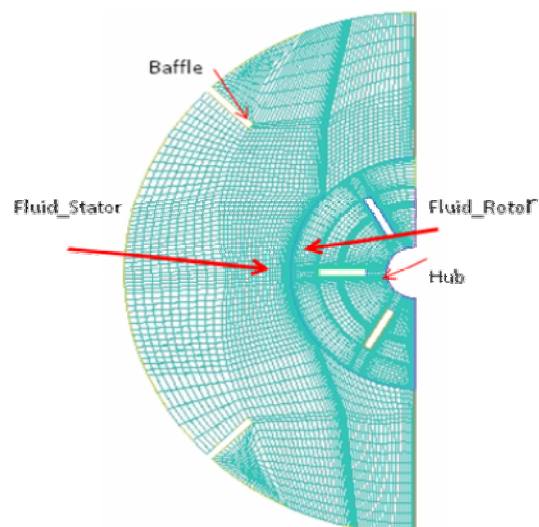**Figure-3(a).** Meshed model of impeller.

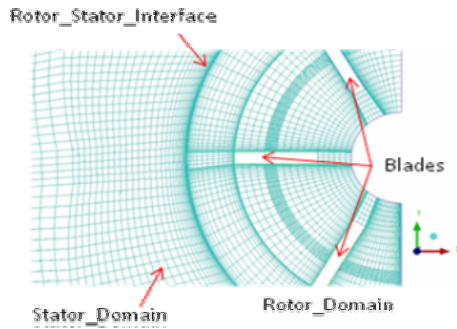**Figure-3(b).** 180° sector volumetric mesh of turbine.

Figure-3(c). Zoomed view of volumetric mesh of fluid domains with interfaces.

GOVERNING EQUATIONS

The three dimensional flow, around the impeller blades in the cylinder, is simulated by producing solution to the appropriate governing equations of conservation of mass, momentum and energy using the ANSYS CFX tool.

Shear stress transport (SST) k-ω model of closure is used to incorporate the turbulence as it has a blending function of k-ε everywhere except k-ω near the wall.

Conservation of mass

$$\nabla(\rho V) = 0 \tag{1}$$

x, y and z momentum

$$\nabla(\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \tag{2}$$

$$\nabla(\rho u V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g \tag{3}$$

$$\nabla(\rho u V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \tag{4}$$

Energy equation

$$\nabla(\rho e V) = -p\nabla V + \nabla(k\nabla T) + q + \square \tag{5}$$

BOUNDARY CONDITIONS

Specification of the boundary conditions is a very crucial part of the simulation. It is necessary to specify appropriate boundary conditions for getting the actual physical process taking place in the Rushton turbine.

The fluid domain inside the vessel is created considering water as the working fluid in ANSYS CFX pre-processor. Bottom wall, cylindrical surface and baffles were modelled as stationary, impermeable walls. Only 180° sector model of the Rushton turbine is considered due to symmetry nature. On the open surfaces of the solution domain symmetry boundary conditions are imposed. For the specification of stationary walls, standard wall functions were used.

The top surface of the liquid was also set as a wall boundary, since it is a closed vessel. Corresponding to the rotational speed, varying angular speeds were assigned to impeller shaft and hub as the boundary condition.

Three fluid-fluid interfaces between the rotating and stationary domains are defined along the left, right side and along the circumference of the impellers, respectively.

GRID INDEPENDENCE STUDY

The CFD analysis for the system was carried out for various numbers of nodal points to predict the dependence of the results on the number of grid points. It was conducted for 311265, 350620,400231 and 450000 nodal points. It was observed that the results were constant after 400000 nodal points. This shows that the results are not much dependent on the grid size above a certain limit of 450000 nodal points for the present study.

RESULTS AND DISCUSSIONS

The development of the flow field around the impeller blade is accessed. The radial velocity, for the given configuration, is computed in the region just outside the blade, behind the blade. The hardware capacity required for the analysis with CPU time and convergence criteria for the various speeds of the Rushton turbine are given in Table-3.

VALIDATION

The numerically predicted results are validated with the available literature of Ranade *et al*. and data points of Schafler *et al*. The results are obtained for z/T=0.329 and r/T= 0.171

The results obtained give a good correlation with very less divergence between the velocities obtained in the angular region of 30 to 60 degrees. The validation of the results is carried out by plotting the non-dimensional radial velocity (Ur/Utip) with that of the experimental data available in the literature as shown in Figure-4. The maximum error percentage for all the values within this range was observed to be around 9.5% except near the edge of the blade where the numerically predicted velocities are much higher than the experimental Methods. The value of Y+ is plotted over the casing and the blade fluid interface as shown in Figure-5(a) and it is found to be less than 10 for incorporating SST - kω model of turbulence.
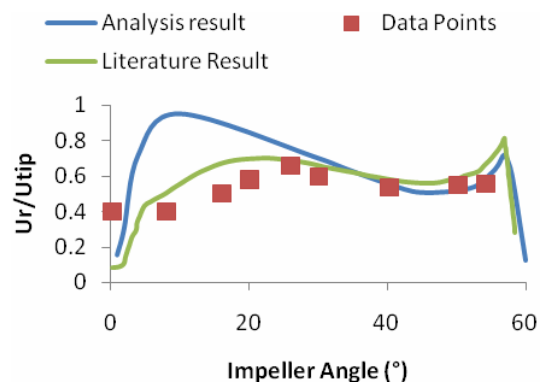

Figure-4. Variation of radial velocity with impeller degree for second blade at 1200 rpm.

www.arpnjournals.com

The deviation in this region due to predictions and experiments may be attributed due to complexity involved in measuring the velocities near these regions.
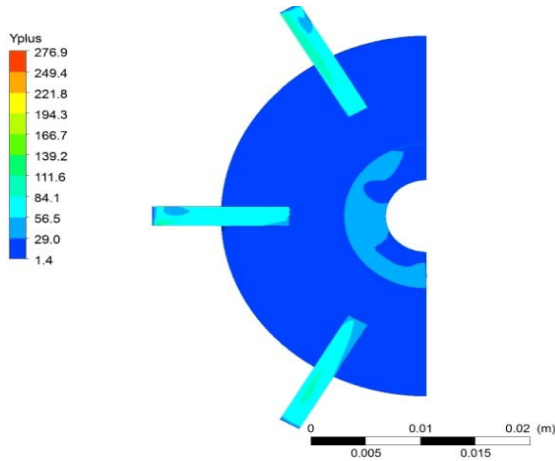

Figure-5(a). Y+ value over the casing and the blades.

The scenario in which the velocity of the impeller is decreased or increased is considered for this present study. The alternate rotation speeds considered in this analysis are 1000 rpm and 1500 rpm. The variation of the non-dimensionalized radial velocity (Ur/Utip) for all the three speeds of the rotor is shown in Figure-5(b).
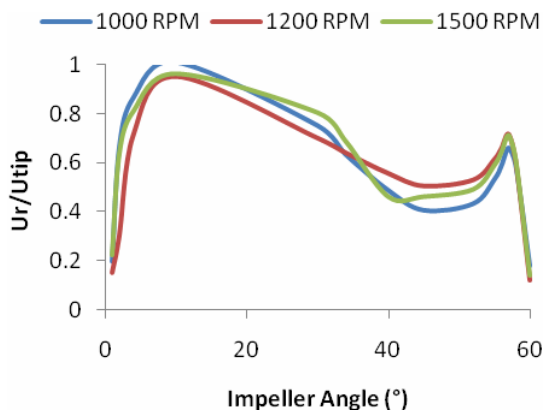

Figure-5(b). Variation of non dimensionalized radial velocity with impeller degree for second blade at various rotation speeds of impeller.

The tip speed for 1500 rpm in 3.927 m/s and for 1000 rpm is 2.618 m/s. The following trend is observed in the radial velocity upon changing the impeller rotation speed as shown in Figure-6. The radial velocity just outside the blade is low and it increases as it proceeds away from the blade radially. A sharp fall in the radial velocity is observed near the region of the neighbouring blade.
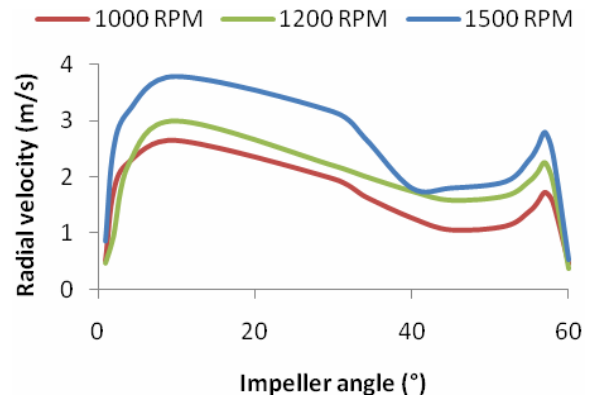

Figure-6. Variation of radial velocity with impeller degree for second blade at various rotation speeds of impeller.

The circulation produced by the baffles is prominent in the stream flow contours of the velocity as shown in Figure-7(a). The otherwise larger single body fluid circulation is presented to be broken down into smaller loops near the baffles.

As the impeller speed is increased an increase in the radial velocity in the flow particles is easily identified. Upon analysing the comparison of various speeds, not a very high change in the flow pattern is observed except that the velocity values have got higher.

The non-dimensional radial velocity for low rotation speed is low for the first half and is higher for the second half in relation to the higher rotation speed. With the increase in the velocity the turbulent kinetic energy is also increased and the flow velocities are also higher.

The contours for the radial velocity for various impeller speeds are presented in Figure-7(b) and Figure-7(c) respectively for 1200 rpm and 1500 rpm respectively. The contours of turbulent kinetic energy for impeller speed of 1000 rpm, 1200 rpm and 1500 rpm are shown from left to right respectively in Figure-8.
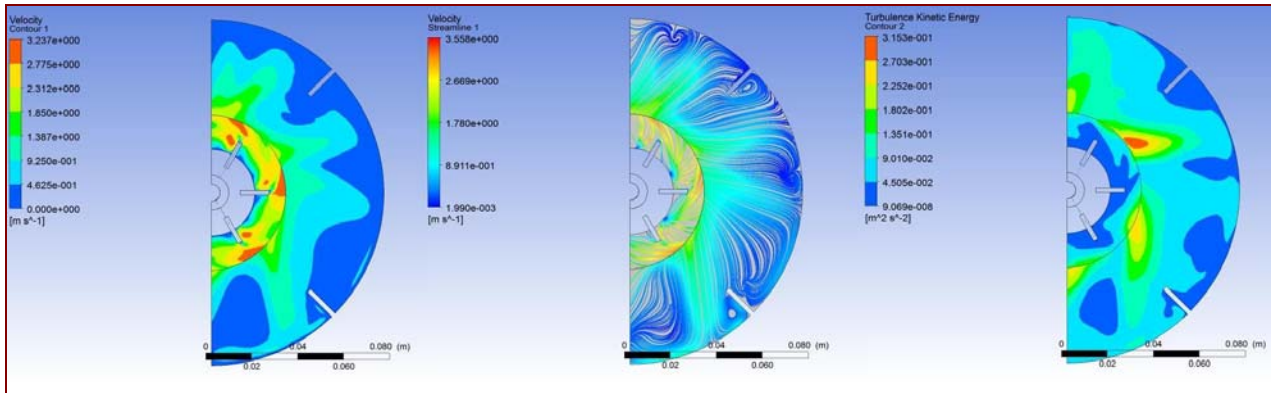
www.arpnjournals.com

Figure-7(a). Contours for radial velocity, streamline radial flow and turbulent kinetic energy contour for 1000 rpm.
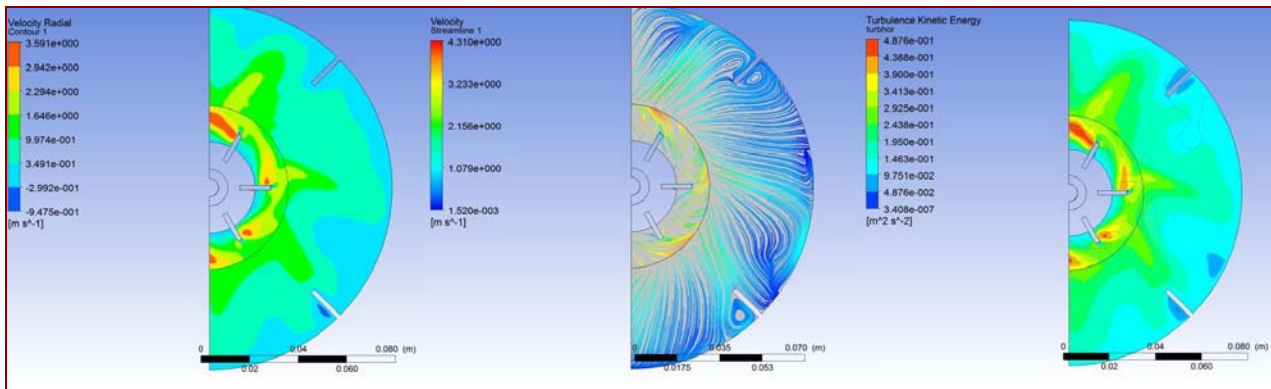

Figure-7(b). Contours for radial velocity, streamline radial flow and turbulent kinetic energy contour for 1200 rpm.
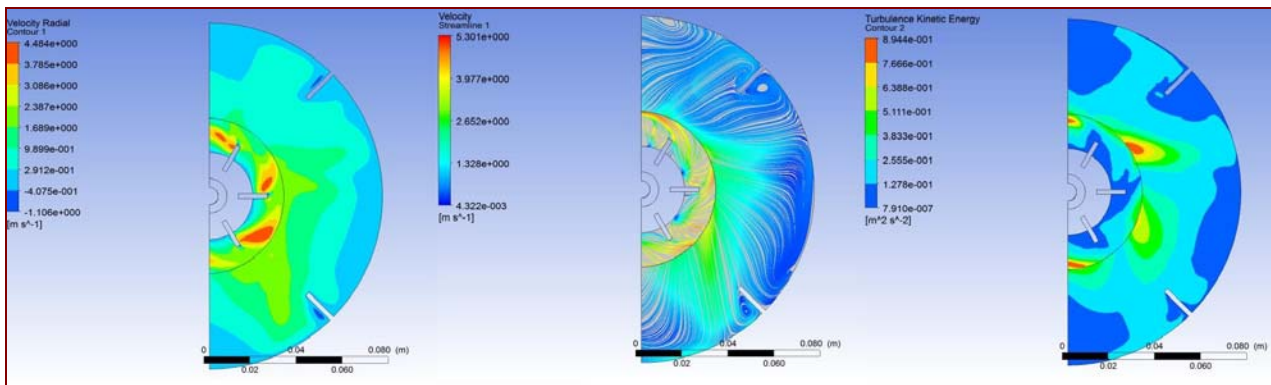

Figure-7(c). Contours for radial velocity, streamline radial flow and turbulent kinetic energy contour for 1500 rpm.
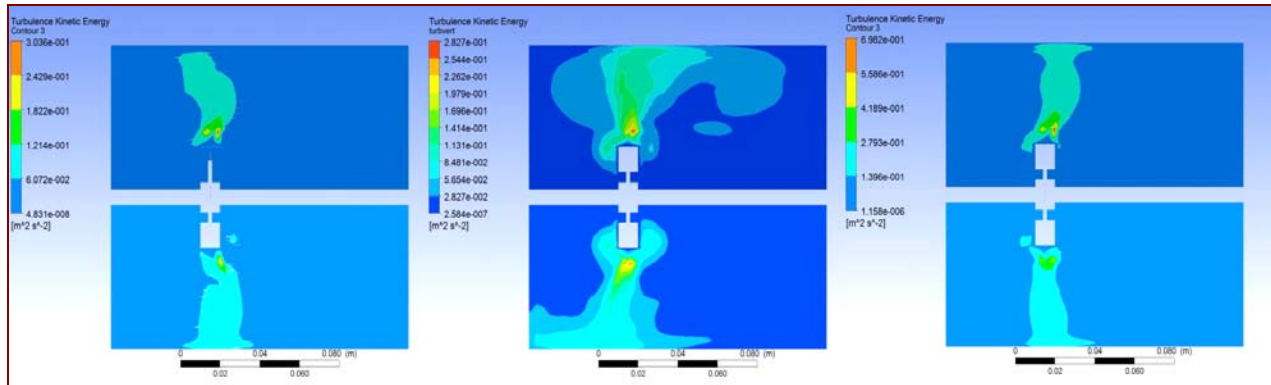
www.arpnjournals.com

Figure-8. Turbulent kinetic energy contour along the vertical plane passing through the blades for 1000rpm, 1200rpm and 1500 rpm, respectively.

Table-3. Hardware requirements and convergence Criteria for various speeds of Rushton Turbine.

S. No.	Speed (RPM)	Hardware configuration	CPU hours Per simulation	Convergence (Mass, momentum and turbulence)	Convergence (Energy)
1	1000	HP Z820 WS System, Intel Xeon E5 2680 (2.7 GHZ, 20 MB Cache, 8 Cores) Intel Chipset C602, 4 GB RAM	9.5	10E-05	10E-07
2	1200	HP Z820 WS System, Intel Xeon E5 2680 (2.7 GHZ, 20 MB Cache, 8 Cores) Intel Chipset C602, 4 GB RAM	9.55	10E-05	10E-07
3	1500	HP Z820 WS System, Intel Xeon E5 2680 (2.7 GHZ, 20 MB Cache, 8 Cores) Intel Chipset C602, 4 GB RAM	9.8	10E-05	10E-07

CONCLUSIONS

The complex turbulent flow characteristics near the Rushton turbine impeller are captured effectively with the CFD method used. The MRF (multiple rotating frames) technique is successfully applied to this case and the results obtained are satisfactory.

The flow contours Figure-7(a) show higher flow rates nearby the impeller. The numerical results presented in this study shows good agreement with the experimental results from the literature for the normalized radial velocity for tip velocity=3.14 m/s (Figure-4) at different angles from the blade of the impeller. It also presents with trailing vortices as mentioned by Ranade *et al*. [16]. This study also presents the results for normalised radial velocities and radial velocities pertaining to tip velocities equal to 3.927 m/s (1500rpm) and 2.618 m/s (1000 rpm) as shown in Figures 5 and 6, respectively the flow field contours and trailing vortices generated are presented in the results of Figure-7(b) and Figure-7(c), respectively.

The turbulence kinetic energy for the planes passing through the centre of the blades is also shown in Figure-8. The results show that as the tip velocity increases, the turbulence near the impeller region increases from 1000 rpm to 1500 rpm.

The spread of turbulent kinetic energy for 1200 rpm is more uniform compared to 1000 rpm and 1500 rpm for the Rushton turbine as shown in Figure-8. This spread of turbulent kinetic energy as well the radial velocity is more uniform at 1200 rpm as seen in Figure-7(b) compared to Figures 7(a) and (c) and hence the optimum speed of the Rushton turbine for this study is concluded with 1200 rpm.

This study also substantiates the claim that the studies relating to mixing in impeller-stirred baffled vessels can be effectively computed using CFD tools for various geometries of the vessel, various rotating speeds as well as for the different types of impellers commercially in use. Also computer simulation provides economic and practical benefits for such studies. It saves the task of experimental procedures as well as construction of new models without being tested. Hence CFD tools are gaining importance in all the fields of application.

REFERENCES

[1] J. Costes and J.P. Couderc. 1988. Study by Laser Doppler anemometry of the turbulent flow induced by a Rushton turbine in a stirred tank: influence of the size of the units. I. Mean flow and turbulence. Chem. Eng. Sci. p. 43.

[2] Fabio Chiti, Serafim Bakalis, Waldemar Bujalski, Mostafa Barigou, Archie Eaglesham and Alvin W. Nienow. 2011. Using positron emission particle tracking (PEPT) to study the turbulent flow in a baffled vessel agitated by a Rushton turbine:

Improving data treatment and validation. Chemical engineering research and design. p. 89.

[3] K.V. Sharp and R.J. Adrian. 2001. PIV study of small scale flow structure around Rushton turbine. AIChE J. 47(4).

[4] J.Y. Luo, R.I. Issa and A.D. Gosman. 1994. Prediction of impeller-induced flows in mixing vessels using multiple frame of reference. Institution of Chemical Engineers Symposium Series. p. 136.

[5] C. Bartels, M. Breuer, K. Wechsler and F. Durst. 2002. Computational fluid dynamics applications on parallel-vector computers: computations of stirred vessel flows. Computational Fluids. p. 31.

[6] Kerim Yapici, Bulent Karasozen, Michael Schafer and Yusuf Uludag. 2008. Numerical investigation of the effect of the Rushton type turbine design factors on agitated tank flow characteristics. Chemical Engineering and Processing. p. 47.

[7] Y. Xu and G. McGrath. 1996. CFD predictions of stirred tank flows. Trans. Inst. Chem. Eng. p. 74.

[8] G. Montante, K.C. Lee, A. Brucato and M. Yianneskis. 2001. Numerical simulations of the dependency of flow pattern on impeller clearance in stirred vessel. Chem. Eng. Sci. p. 56.

[9] R. Zadghaffari, J.S. Moghaddas and J. Revstedt. 2010. Large-eddy simulation of turbulent low in a stirred tank driven by a Rushton turbine. Computers and Fluids. p. 39.

[10] Harminder Singh A, David F. Fletcher b and Justin J. Nijdama. 2011. An assessment of different turbulence models for predicting flow in a baffled tank stirred with a Rushton turbine. Chemical Engg. Science. p. 66.

[11] Aoyi Ochieng, Maurice S. Onyango, Anil Kumar, KirimiKiriamiti and Paul Musonge. 2008. Mixing in a tank stirred by a Rushton turbine at a low clearance Chem. Engg. and Processing. p. 47.

[12] Mahsa Taghavi, Ramin Zadghaffari, Jafarsadegh Moghaddas and Yousef Moghaddas. 2011. Experimental and CFD investigation of power consumption in a dual Rushton turbine stirred tank. Chemical Engineering Research and Design. 8(9).

[13] H.S. Yoon, D.F. Hill, S. Balachandar, R.J. Adrian and M.Y. Haa. 2005. Reynolds number scaling of flow in a Rushton turbine stirred tank. Part I-Mean flow, circular jet and tip vortex scaling. Chemical Engineering Science. p. 60.

[14] Marina Campolo, Fabio Sbrizzai and Alfredo Soldati. 2003. Time-dependent flow structures and Lagrangian mixing in Rushton-impeller baffled-tank reactor. Chemical Engineering Science. p. 58.

[15] M. Schafer, M. Hofken and F. Durst. 1997. Detailed LDV measurements for visualization of the flow field within a stirred-tank reactor equipped with a Rushton turbine. Institution of Chemical Engineers. p. 263.

[16] V.V. Ranade and S.M.S. Dommeti. 1996. Computational snapshot of flow generated by axial impellers in baffled stirred vessels. Trans. Inst. Chem. Eng. p. 74.