



PERFORMANCE STUDY OF MIXING AGITATOR USING COMPUTATIONAL FLUID DYNAMICS

M. Sanju Edwards* & V. G. Ganesan**

* PG Student, Department of Mechanical Engineering, Easwari Engineering College, Ramapuram, Tamilnadu

**Assistant Professor, Department of Mechanical Engineering, Easwari Engineering College, Ramapuram, Tamilnadu

Abstract:

Mixing in stirred tanks is one of the most common unit operations in process industry. Mixing is usually carried out at low to moderate Reynolds number. Impellers are mainly classified based on fluid flow. 1. Radial flow 2. Axial flow. In the present work the base model of the rushton agitator will be taken to study the mixing performance using CFD and compare it with literature by plotting performance parameters (U_r/U_{tip} vs r/R), ($2z/b$ vs U_x/U_{tip}) and (Q_r vs r/R). Once after validating and comparing the performance parameters from CFD simulation with reference literature, the design modification will be considered. The CFD simulation of the stirring chamber will be carried out with modified agitator by replacing the rushton agitator. The CFD simulation will be carried out for different rotational speeds 225,300 and 400 rpm respectively. Comparative study between base design and proposed design will be studied based on radial flow rate, power number and flow number.

Index Terms: Stirred Tanks, Process Industry, Impeller, Fluid Flow & Rushton Agitator

1. Introduction:

A stirred tank reactor is the simplest type of reactor. It is composed of a reactor and a mixer such as stirrer, a turbine wing or a propeller. This type is useful for substrate solutions of high viscosity and for immobilized enzymes with relatively low activity. However, a problem that arises is that an immobilized enzyme tends to decompose upon physical stirring. This type is suitable for the production of rather small amounts of chemicals or for mixing purpose.

Components of stirred tank

- Stainless steel vessel
- Shaft with propeller blade
- Liquid substance for mixing purpose

Tanks stirred by either a mechanical means or by a fluid jet are widely used in many industrial processes. These processes include the chemical, petrochemical, oil and metallurgical industries. Stirred tanks are used in diverse processes such as blending, dispersing, emulsifying, suspending and enhancing heat and mass transfer. Consequently, a very wide range of stirred tanks is available to suit various applications. The details of the flow and the extent of mixing in a stirred tank are of direct interest in most industrial applications. The design and scale-up of stirred tanks and the quantification of mixing in them have been traditionally tackled by developing empirical design equations mainly due to the complexity of the fluid dynamics of mixing. For example, for a given unit, the degree of mixing is deduced by analyzing the residence-time distributions of a tracer. Although this approach has proven to be satisfactory for many applications, it is rather limited because it neglects the complexity of flow in most mixing applications. Moreover the empirical equations are usually highly experiment specific and seldom contribute to the development of theory. Recently computational fluid dynamics (CFD) and advanced experimental techniques such as laser-Doppler velocimetry (LDV) have been increasingly used to obtain better understanding of the

mixing process, including detailed knowledge of the flow characteristics. Such a detailed understanding of the process is essential for equipment design and selection. Computer simulation of turbulent flow phenomena has been successfully applied to many industrial applications. Patterson (1975) described the principles of applying.

The impellers are mainly classified based upon the fluid flow. They are radial and axial flow. The Rushton turbine is come under radial flow. The main disadvantage of the Rushton turbine is high power number (PO), because of the fluid resistance. The aim of this work is to investigate the effect of the shaft eccentricity on the hydrodynamics of baffled stirred vessels. The difference between coaxial and eccentric agitation is studied by calculating flow pattern and power number of the stir tank for the different rpm for 225,300 and 400 of the impeller blade by experimental and CFD analysis.

2. Experiment:

The design and the dimensions of the stirred tank with two Rushton impellers of standard geometry, which were used in this work, are shown in Figure 1. The stirred tank consists of a flat bottomed glass cylinder (refractive index 1.47 at a wavelength of 587 nm) an inner diameter T of 0.30m. Four vertical baffles are symmetrically placed around the tank wall, each with a width of one-tenth of the tank diameter ($b_{\text{baffle}} = 0.1T$). The Rushton stirrer with two impellers, which is of standard design with a diameter (D) equal to one-third of the tank diameter ($D/T=1/3$), is located with a clearance from the tank bottom of about half of the tank diameter ($C1=0.55T$). The upper impeller is placed $\Delta C=0.7T$ above the lower one. A pump drives the turbines and the stirring speed is measured using a calibrated digital oscilloscope. In order to reduce optical refractive index effects at the cylindrical surface of the tank, it is placed in a square glass vessel. The stirred tank is filled with tap water as the main continuous phase fluid, the surface of which is $C2=0.55T$ above the upper impeller. The square vessel is also filled with water to reduce light refraction at the interface. For more details on the geometry of the stirred tank used in this study see Guillard et al.[3]. A double-cavity 2×25 mJ Nd:Yag (Continuum) pulsed laser is used to produce a beam at a wavelength of 532 nm.

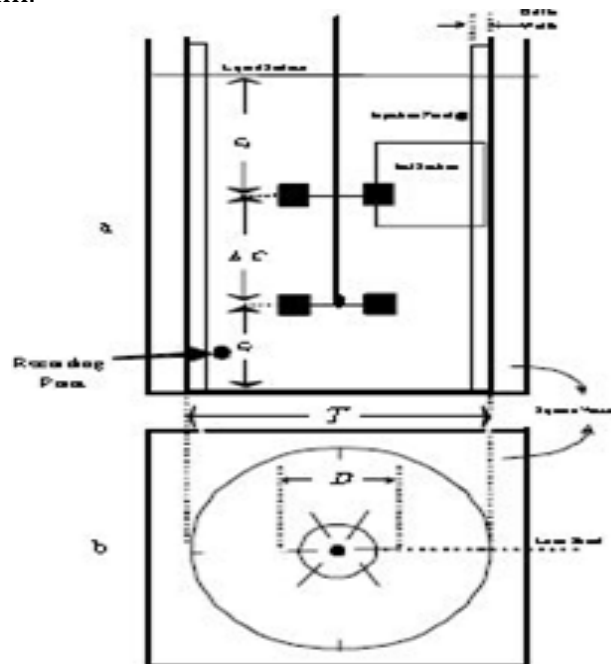


Figure 1. Stirred tank and impellers:
a) side view, b) top view.

The laser beam passes through a plano-concave lens to produce a two dimensional vertical sheet of light (2 mm thick). Two series of experiments were performed: i) velocity measurements by using of PIV, and ii) concentration measurements of the determining tracer by using of a CCD camera (PLIF mode). In each series 3 different rotational speeds of the impellers: 225, 300 and 400 rpm, were employed. For the determination of the average velocity and mixing characteristics, the CCD camera captured 500 instantaneous images in each case. The average of the 500 images was used in the calibration of the PLIF measurement. The calibration procedure is described by Moghaddas et al. [4].

3. CFD Method:

Three-dimensional CFD code Fluent, version 6.1, a finite volume based 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. A number of strategies can be used to deal with the movement of the impeller blades as mentioned before. In this study, the MRF (Multiple Reference Frame) solution was used as a starting point. The simulation was then switched to unsteady SM (Sliding Mesh) model with time step set to 0.001 sec and second order upwind scheme for discretization and SIMPLE algorithm for pressure velocity coupling were used. During the unsteady computations to judge the convergence, volume integral of kinetic energy in the tank was calculated. When the pseudo-steady state was reached (after 49s of real time), the computations of tracer distribution in the vessel were solved together with the flow equations. A computational grid consisting of 610000 cells and structured grids composed of non-uniformly distributed hexahedral cells were made in Gambit 2.1 pre-processor. Water at 25°C was used as the test fluid ($\rho=103 \text{ kgm}^{-3}$, $\mu=10^{-3} \text{ Pa s}$). The simulation was run in several impeller rotational speeds in turbulence domain with an initial condition of zero velocity at all grid nodes.

4. Result and Discussion:

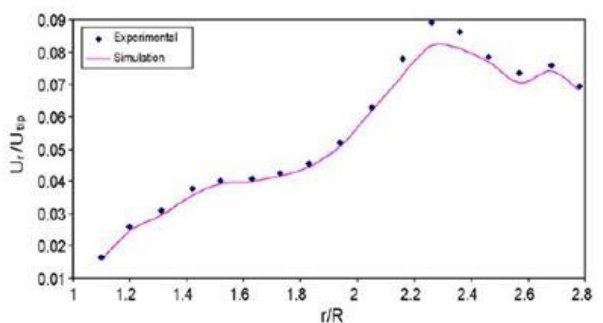
4.1 Simulation Case for Design Validation with Literature:

- Twin Ruston Agitator Simulation
- For Chamber Fluid – Water
- Rotational Speeds -225 rpm, 300 Rpm And 400 rpm
- For Chamber Fluid - Glycerine
- Rotational Speed - 225 rpm, 300 Rpm And 400 rpm

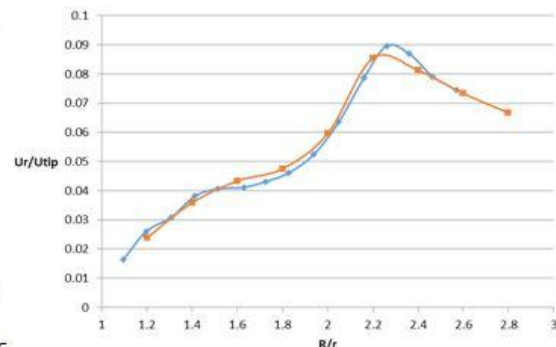
4.2 Design Validation with Experiment from Literature:

R. Zadghaffari, 2009 – Double Rushton Impeller Experimental non-dimensional velocity curve with water as chamber fluid

Simulation curve validation with experiment – Fluid (water)



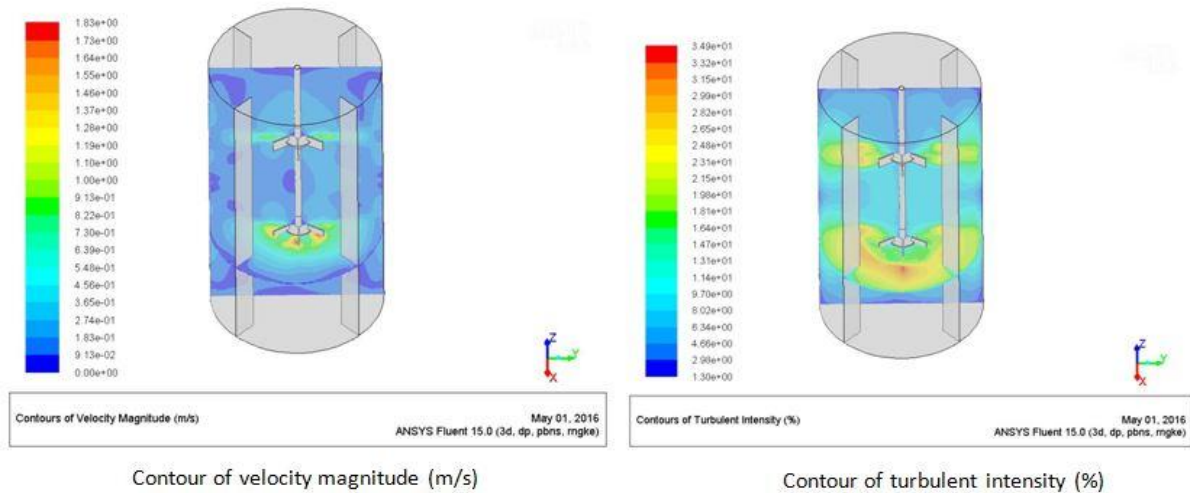
Normalised radial mean velocity, RPM=300, $2z/b = -2.36$



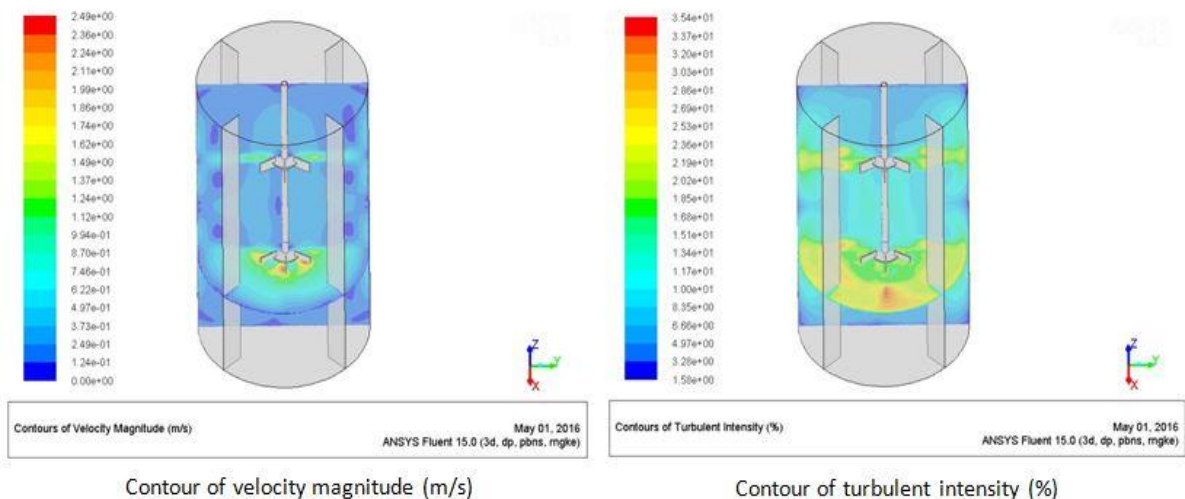
Double Rushton - Experiment
CFD Simulation-Present work

4.3 Simulation Results for Water:

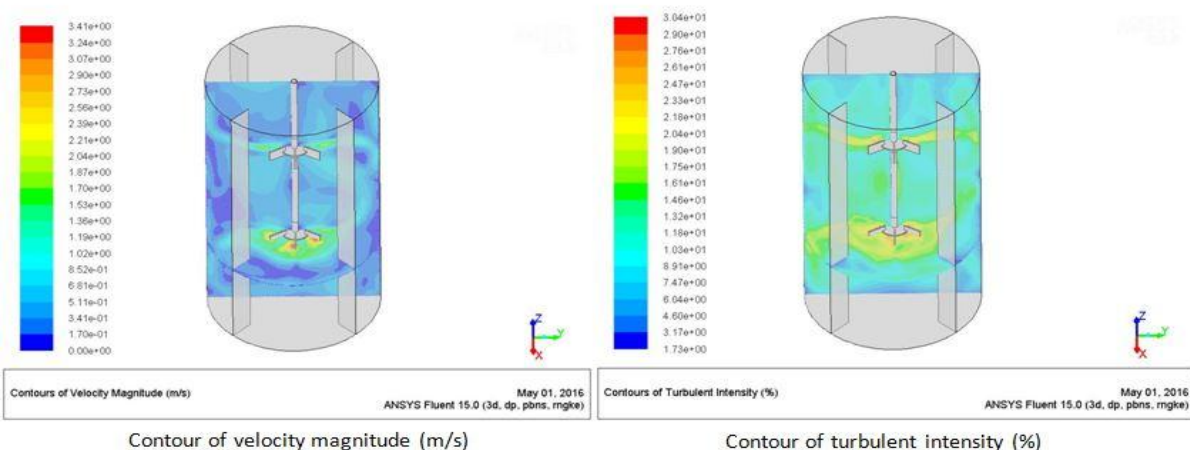
4.3.1 Simulation Results for Water as Chamber Fluid and Rotational Speed=225 rpm:



4.3.2 Simulation Results for Water as Chamber Fluid and Rotational Speed=300 rpm:

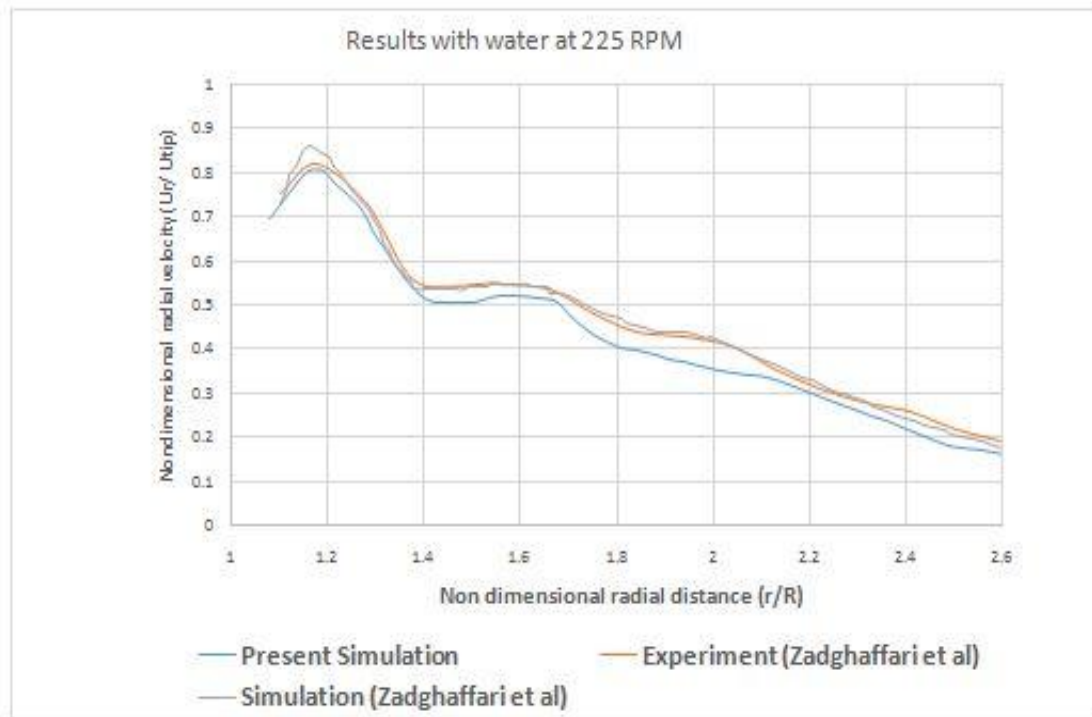


4.3.3 Simulation Results for Water as Chamber Fluid and Rotational Speed=400 rpm:

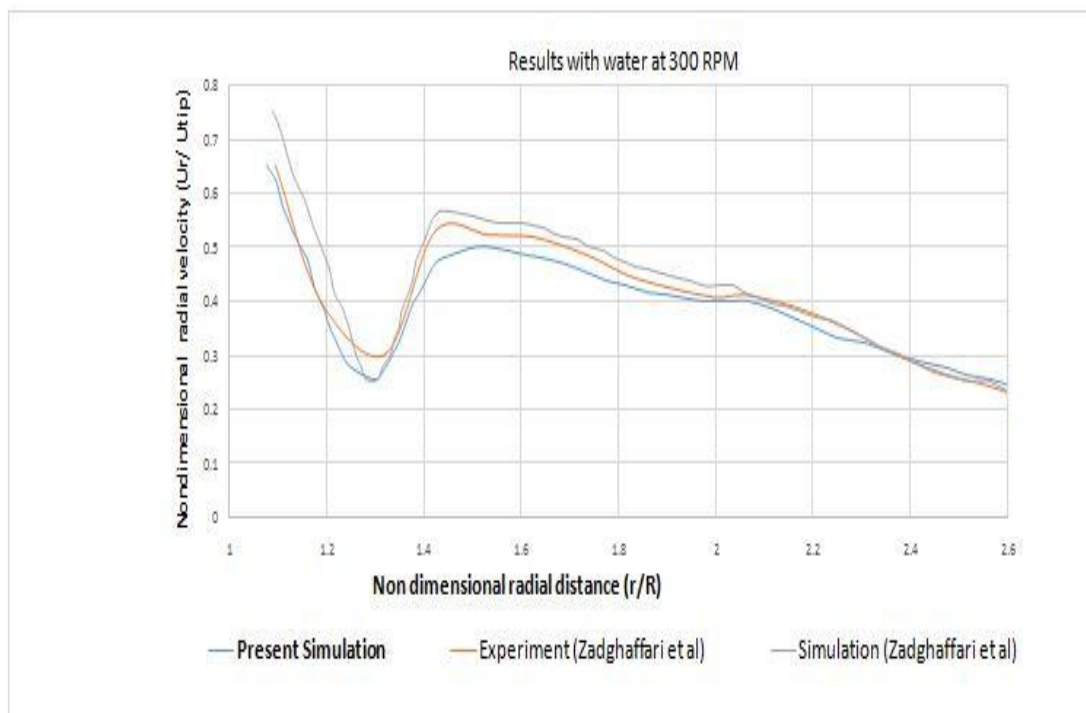


4.4 Velocity Curve for Simulation Curve of Water:

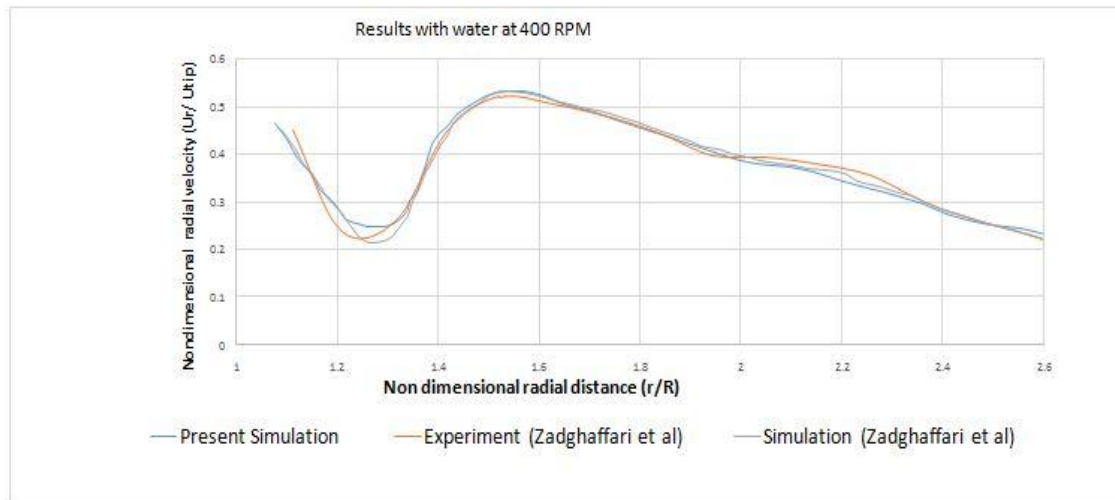
4.4.1 Velocity for Simulation Case -Chamber Fluid as Water with Stirrer Rotational Speed=225 rpm:



4.4.2 Velocity for Simulation Case -Chamber Fluid as Water with Stirrer Rotational speed =300 rpm:

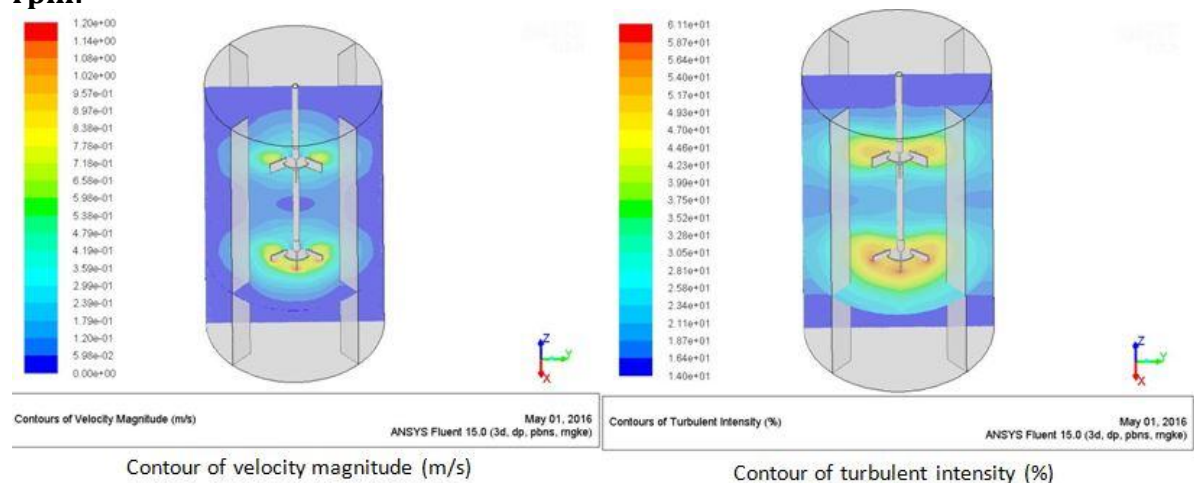


4.4.3 Velocity for Simulation Case -Chamber Fluid as Water with Stirrer Rotational Speed =400 rpm:

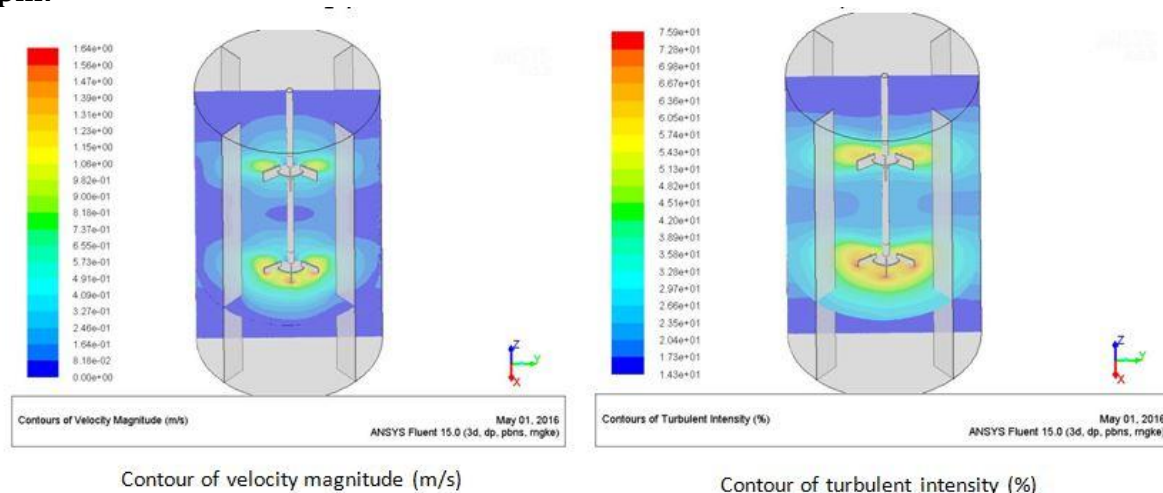


4.5 Simulation Results for Glycerine:

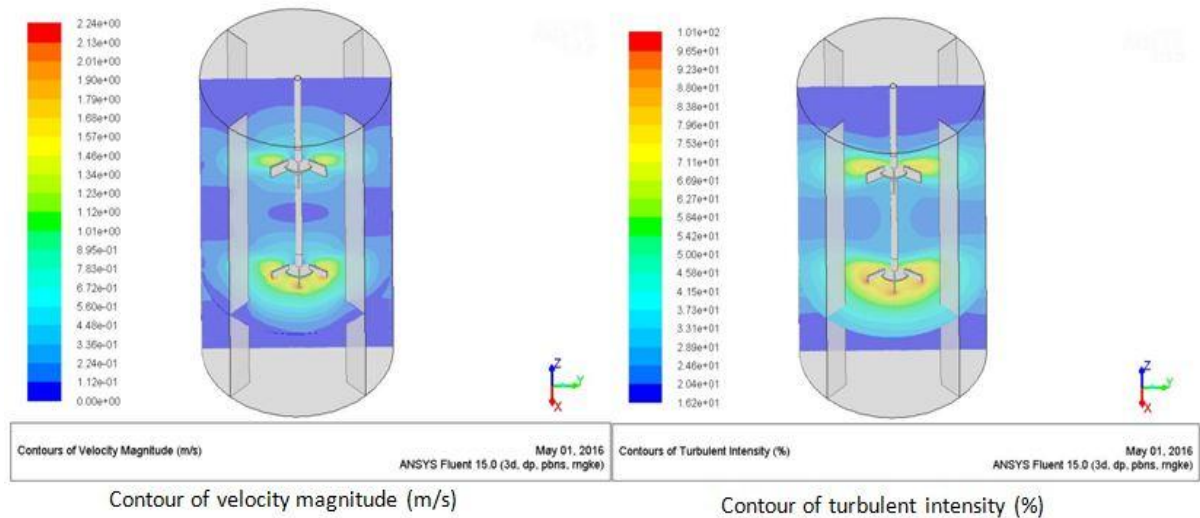
4.5.1 Simulation Results for glycerine as chamber fluid and rotational speed =225 rpm:



4.5.2 Simulation Results for glycerine as chamber fluid and rotational speed =300 rpm:

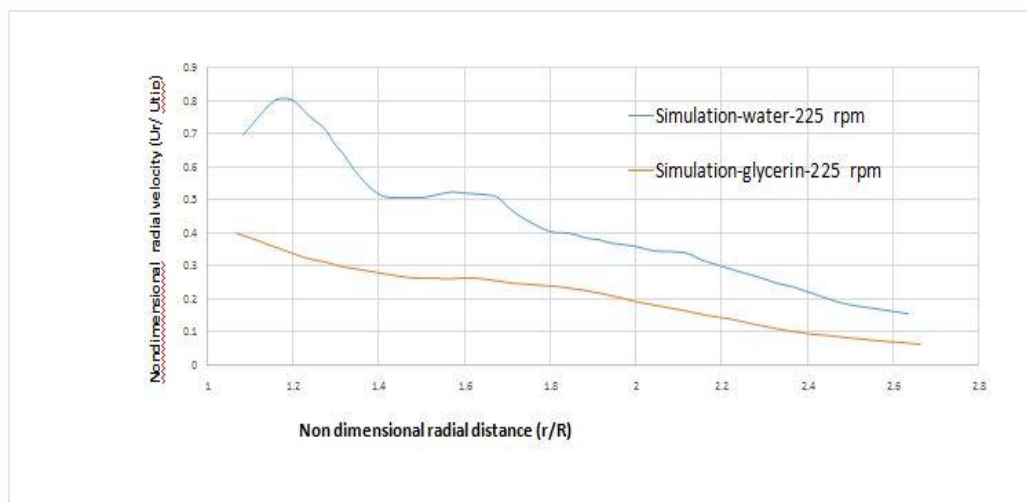


4.5.3 Simulation Results for glycerine as chamber fluid and rotational speed =400 rpm:

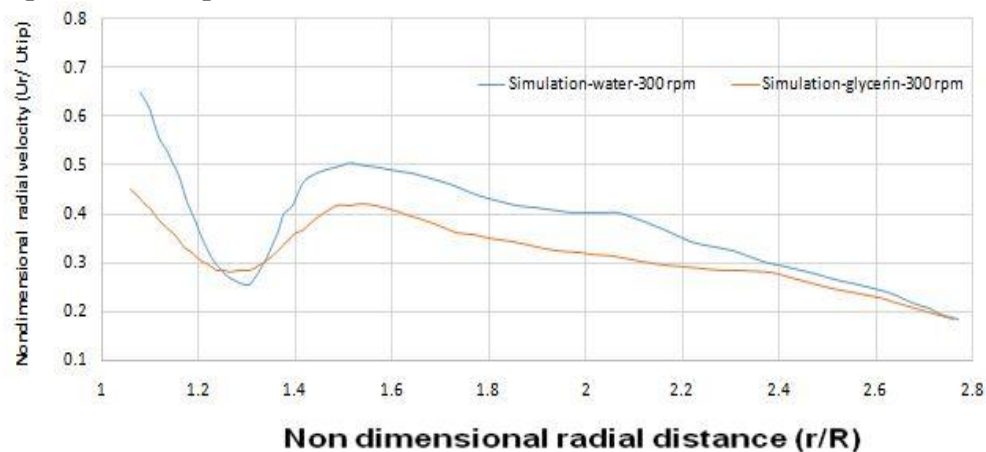


4.6 Velocity Curve for Simulation Curve of Glycerine:

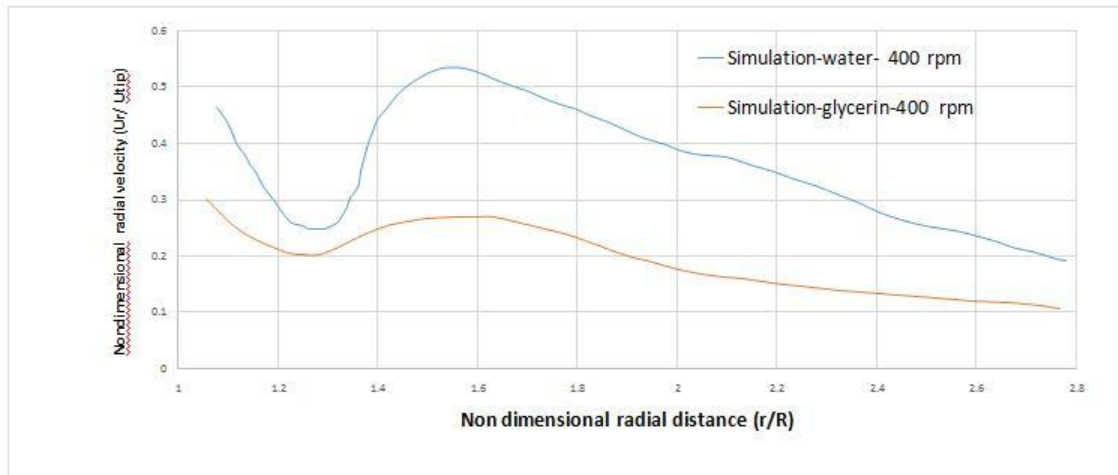
4.6.1 Velocity for Simulation Case-Chamber fluid as glycerine with stirrer rotational speed =225 rpm:



4.6.2 Velocity for Simulation Case -Chamber fluid as glycerine with stirrer rotational speed =300 rpm:



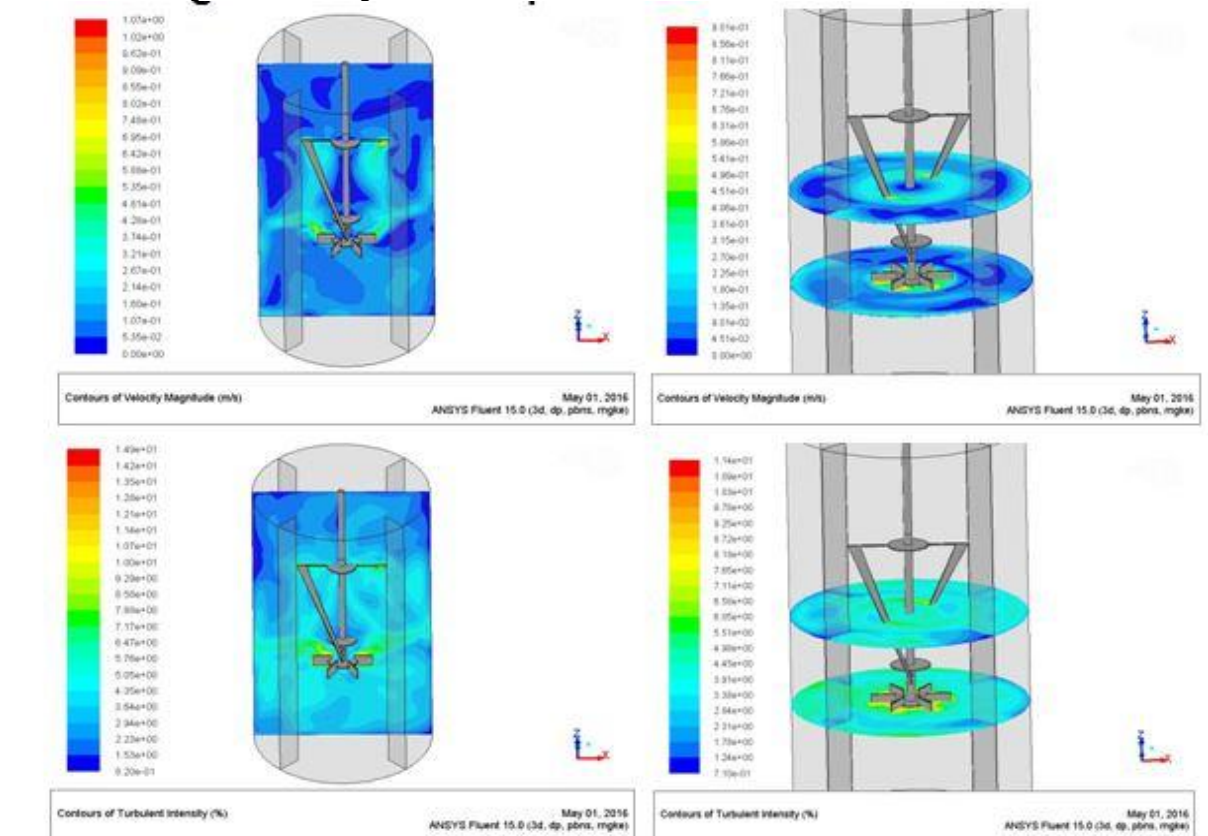
4.6.3 Velocity for Simulation Case-Chamber fluid as glycerine with stirrer rotational speed=400 rpm:



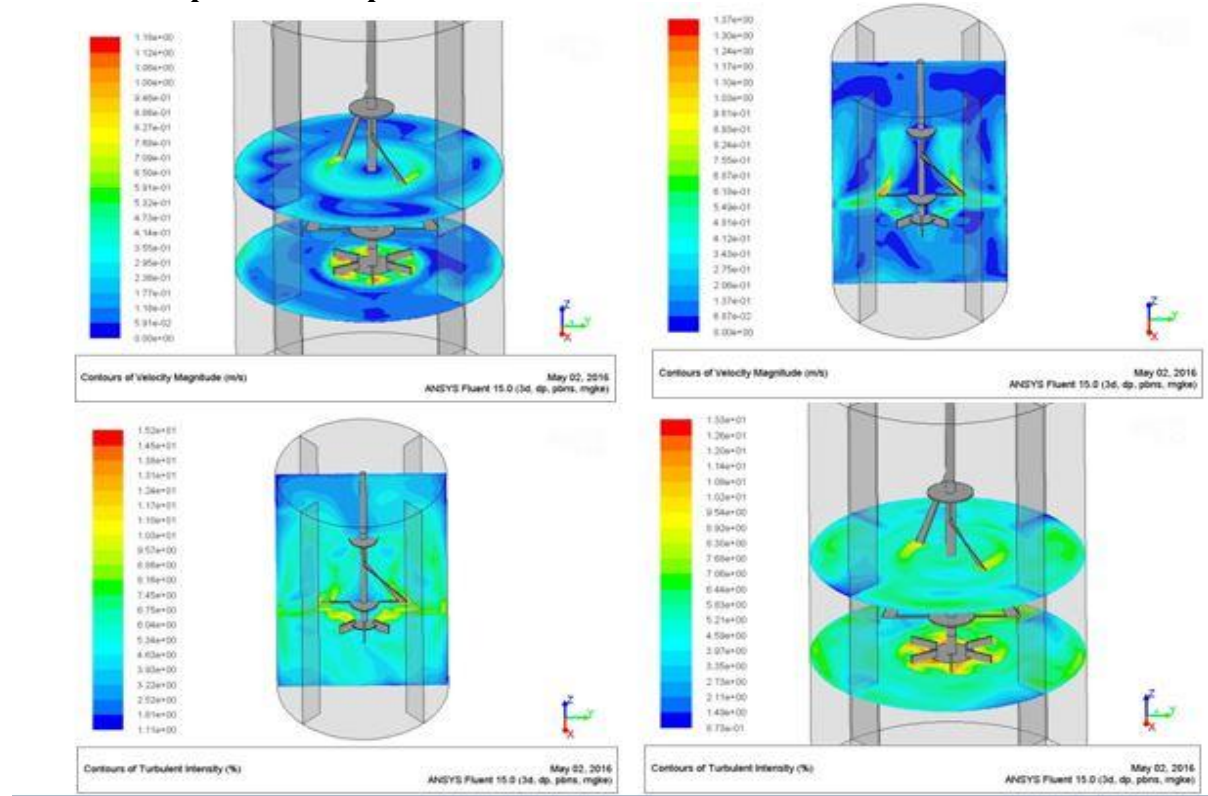
4.7 Simulation Case for Modified Design Rushton Turbine Replaced with Twisted Twin Blade Agitator:

- Rushton turbine-2 replaced with twisted twin blade agitator
- For Chamber Fluid - Water
- Rotational speed - 110 rpm, 150 rpm and 200 rpm
- For Chamber Fluid - Glycerine
- Rotational speed - 110 rpm, 150 rpm and 200 rpm

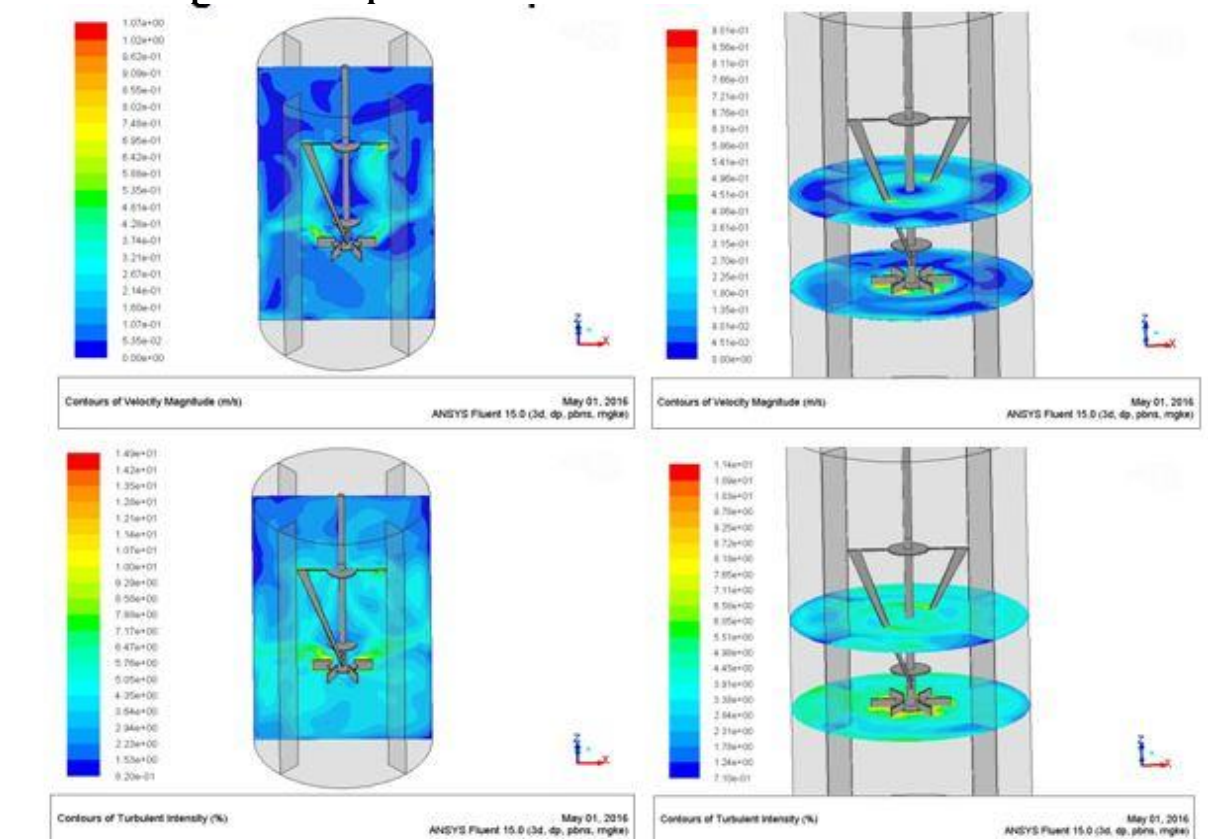
4.7.1 Simulation Case for Modified Design for Chamber Fluid Water with Rotational Speed=110 rpm



4.7.2 Simulation Case for Modified Design for Chamber Fluid Water with Rotational Speed =150 rpm

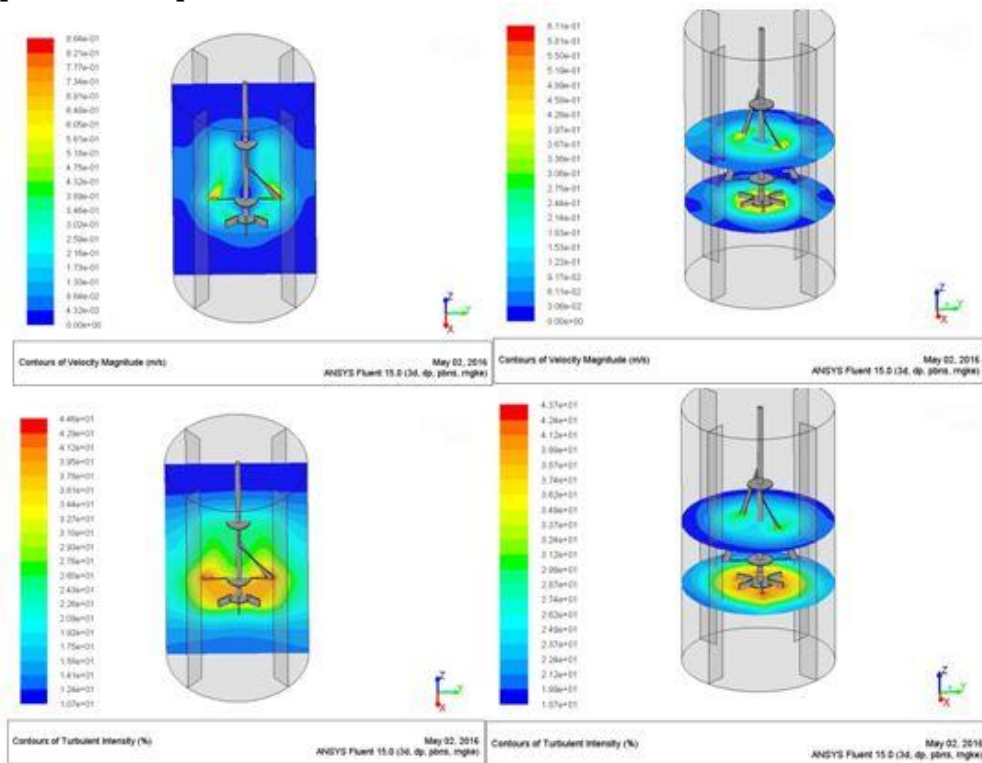


4.7.3 Simulation Case for Modified Design for Chamber Fluid Water with Rotational Speed =200 rpm

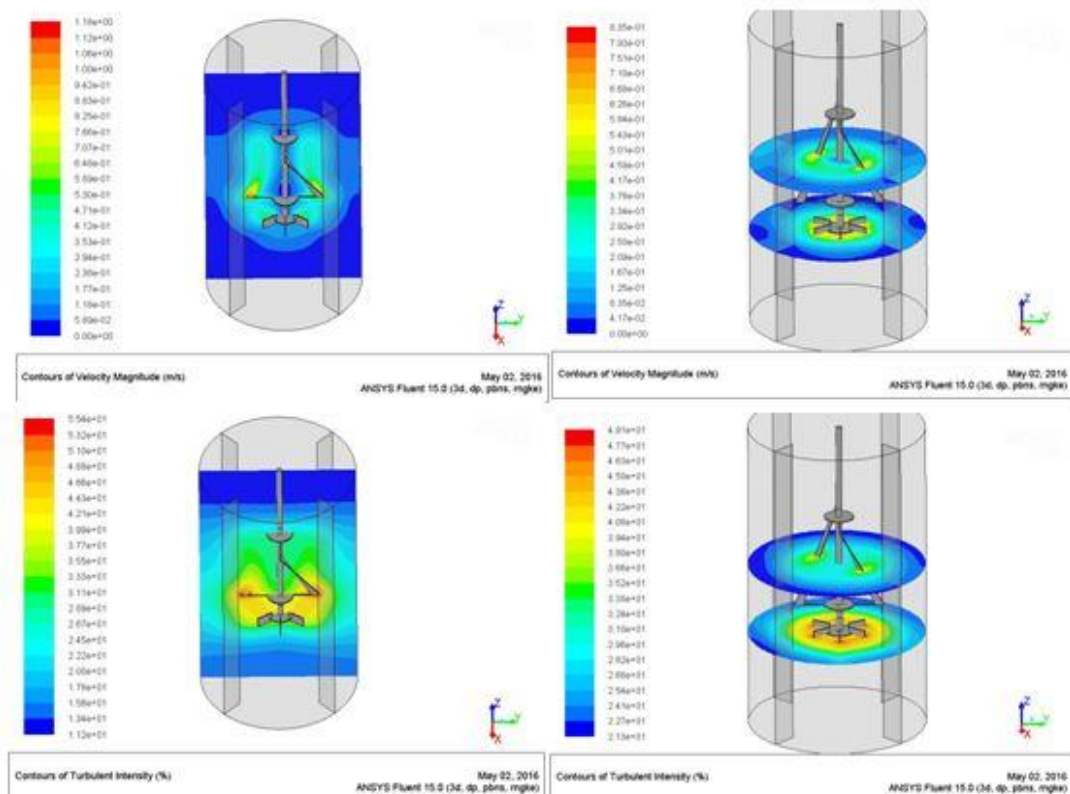


4.8 Simulation Case for Glycerine:

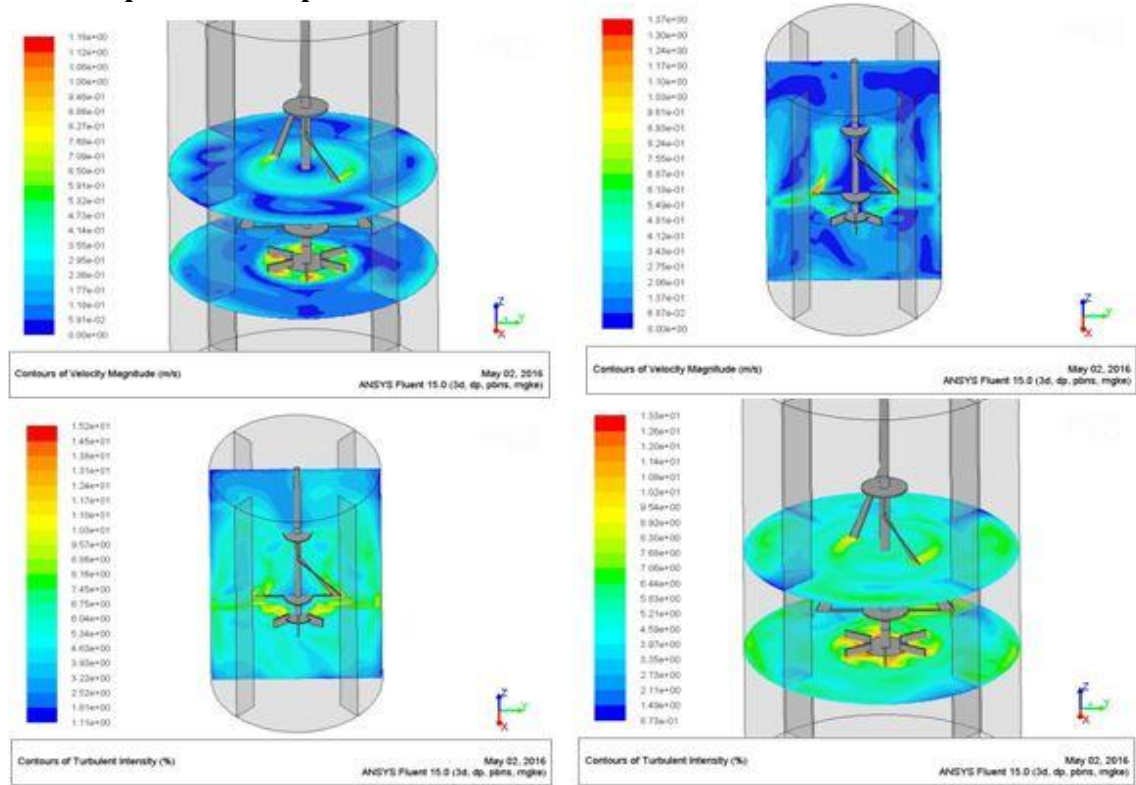
4.8.1 Simulation Case for Modified Design for Chamber Fluid Glycerine with Rotational Speed =110 rpm



4.8.2 Simulation Case for Modified Design for Chamber Fluid Glycerine with Rotational Speed =150 rpm



4.8.3 Simulation Case for Modified Design for Chamber Fluid Glycerine with Rotational Speed =200 rpm



5. Conclusion:

In the present work, the mixing field in a baffled tank stirred by two flat six-blade Rushton turbines was predicted using the CFD code, Fluent 6.1, at three different impeller rotational speeds: 225,300 and 400 rpm, for the single phase mixing system. An LES subgrid scale model was used as a turbulent model and the sliding mesh method was used for counting the impeller rotation. For validation of the simulation results the mixing field was experimentally analysed using the particle image velocimetry (PIV) and planar laser-induced fluorescence (PLIF) measurement techniques. Based on the PIV results, the pumping capacity of the systems was calculated. A considerable reduction in mixing time was achieved by increasing the impeller speed. Reasonable agreement between the experimental and the simulation results was obtained. The satisfactory comparisons indicate the potential usefulness of this CFD approach as a computational tool for designing stirred reactors.

6. References:

1. Aubin J., Mavros P., Fletcher D.F., Bertrand J. and Xuereb C. (2001). 'Effect of axial agitator configuration (up-pumping, down-pumping, reverse rotation) On flow patterns generated in stirred vessels' Trans IChemE, Vol 79,Part A, pp.845-856.
2. Chapple D., Kresta S.M., Wall A. and Afacan A. (2002). 'The effect of Impeller and Tank Geometry on Power number for a pitched blade turbine' Trans IChemE, Chemical Engineering Research and Design Vol 80, Part A pp.364-372.
3. Khopkar A.R., Mavros P., Ranade V.V. and Bertrand J. (2004). 'Simulation of flow generated by an Axial-flow impeller Batch and continuous operation' Trans IChemE, Part A, Chemical Engineering Research and Design, 82(A6), pp.737-751.
4. Foucault S., Ascanio G., Tanguy P.A. (2005). "Power characteristics in coaxial mixing: Newtonian and non-Newtonian fluids", Ind. Eng. Chem. Res., 44, 5036-5043.

5. Mansour Jahangiri (2006). 'Fluctuation Velocity for Non-Newtonian Liquids in Mixing Tank by Rushton Turbine in the Transition Region' Iranian Polymer Journal Vol 15 (4),pp. 285-290.
6. Montante G., Bakker A., Paglianti A., Magelli F. (2006). 'Effect of the shaft eccentricity on the hydrodynamics of unbaffled stirred tanks' Elsevier, Chemical Engineering Science 61, pp.2807 – 2814
7. T. Kumaresan, Jyeshtharaj B. Joshi (2006). Effect of impeller design on the flow pattern and mixing in stirred tanks, Chemical Engineering Journal 115 173–193.
8. Elmabruk A. Mansur, Wang Yun-dong, Dai You-yuan (2008). 'Computational Fluid Dynamic Simulation of Liquid–Liquid Mixing in a Static Double-T-shaped Micromixer' The Chinese Journal of Process Engineering Vol.8 No.6, pp.1080-1084.
9. Bart C.H. Venneker, Jos J. Derksen, Harry E.A. Van den Akker (2010). 'Turbulent flow of shear-thinning liquids in stirred tanks—The effects of Reynolds number and flow index' Elsevier, chemical engineering research and design 88, pp.827–843.