LAMINAR FLOW AND HEAT TRANSFER IN STIRRED VESSEL USING CFD

Thesis submitted in partial fulfillment of the requirements for the award of the degree of

MASTER OF ENGINEERING

IN

THERMAL ENGINEERING

Submitted by

JAGJIT SINGH

(821183003)

Under the supervision of

Dr. Avinash Chandra
Assistant Professor
Department of Chemical Engineering
Thapar University, Patiala

Dr. Vikas Kumar Sangal
Assistant Professor
Department of Chemical Engineering
Thapar University, Patiala

MECHANICAL ENGINEERING DEPARTMENT

THAPAR UNIVERSITY, PATIALA-147004, INDIA

JULY 2014

DECLARATION

I hereby certified that the work which is being presented in dissertation entitled, " **LAMINAR FLOW AND HEAT TRANSFER IN STIRRED VESSEL USING CFD**" in partial fulfillment of requirements for the award of the degree of Master of Engineering in Mechanical Engineering with specialization in **THERMAL ENGINEERING** submitted in **Mechanical Engineering Department** of Thapar University, Patiala, is an authentic record of my own work carried out under the supervision of **Dr. Avinash Chandra** and **Dr. Vikas Kumar Sangal**. The work refers other researcher's works which are duly listed in the references section.

The matter presented in this thesis has not been submitted for the award of any other degree of this or any other university.

Date: 17/7/14
Place: Patiala

JAGJIT SINGH
(821183003)

There is to certify that above statement made by the candidate is correct and true to the best of our knowledge.

18/07/14

Dr. Avinash Chandra
Assistant Professor
Department of Chemical Engineering
Thapar University, Patiala

18/07/14

Dr. Vikas Kumar Sangal
Assistant Professor
Department of Chemical Engineering
Thapar University, Patiala

Countersigned by

Dr. Ajay Batish
Professor& Head of Department
Department of Mechanical Engineering
Thapar University, Patiala

Dr. S.K. Mohapatra
Dean of Academic Affairs
Thapar University, Patiala

I dedicate this thesis to my

PARENTS & SISTER

whose unconditional love has always supported me.

ACKNOWLEDGEMENT

My years of Post graduate study are finally nearing completion. This is the opportune moment to acknowledge all who have crossed my path during the journey and contributed directly or indirectly to reaching the light at the end of the tunnel.

Undoubtedly, my supervisors, **Dr. AVINASH CHANDRA** and **Dr. VIKAS KUMAR SANGAL**, Assistant Professors of Chemical Engineering Department, Thapar University, Patiala, deserve to be at the top of the list. It was an excellent opportunity to work under the tutelage of two stalwarts of Chemical engineering. These years have been helpful for my academic progress. I sincerely thank them for their kind guidance, helpful suggestions, encouragement. The fruitful discussions with them throughout the research work were a great help for the successful completion of the present work.

It is a great pleasure to thank the authorities of Thapar University, Patiala for providing the opportunity to carry out my research work successfully.

I owe sincere and earnest thankfulness to my family who enthused, encouraged and fully supported me for every trial comes in my life. I acknowledge my sincere thanks to all my friends for providing the companionship and making my stay at Thapar University, Patiala pleasant and joyful.

Last but not the least, To God the father of all, I thank for the strength that keeps me standing and for the hope that keeps me believing that this affiliation would be possible and more interesting.

JAGJIT SINGH

ABSTRACT

Agitated vessel have been widely used for blending and mixing operations in process industries such as soap, oil, cosmetic, paint industries, etc. The mixing and blending operations are usually performed at elevated temperature. Hence, the heat transfer is very important phenomenon in agitated vessels. The laminar flow and heat transfer in an agitated vessel were studied numerically using commercial CFD software ANSYS FLUENT. The study has been performed at Reynolds number (Re=20) and for the Prandtl number (Pr) ranges from 0.71-50. The total six (Pr =0.71, 7, 20, 30, 40, 50) intermediate Prandtl number has been chosen in between 0.71-50. An agitated cylindrical vessel equipped with four blade pitched turbine. The vessel has flat top and bottom surface and all the vessel surfaces are kept at uniform constant wall temperature. The impeller was placed concentrically in the vessel. The segregated solver was used to obtain desired simulation results. After validation of our numerical methodology, the results for steady laminar flow and heat transfer have been presented. The results show that the velocity gradient is more near to the impeller region then near to vessel wall. Furthermore, heat transfer is shows that the temperature in the impeller region is almost constant and the steep temperature gradient near to vessel wall. The heat transfer coefficients are presented in terms of average Nusselt number (Nu). Overall the heat transfer increases with increase in Prandtl number at constant value of Reynolds number and a correlation is proposed to get the intermediate values of Prandtl number.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF FIGURES

NOMENCLATURE

Pr	*Prandtl number*
H	*Impeller blade height, m*
C	*Impeller off-bottom clearance, m*
Cp	*Specific heat capacity, J/ (kg·K)*
D	*Impeller diameter, m*
T	*Diameter of the vessel, m*
g	*Gravitational acceleration, m/s2*
h	*Heat transfer coefficient, W/(m2K)*
H_l	*Liquid depth in the vessel, m*
k	*Thermal conductivity, W/(m-K)*
n	*Impeller blade number*
N	*Impeller rotational speed, rev/s*
Np	*Power number of impeller, (-)*
Nu	*Nusselt number, (-)*
P	*Pressure, N/m2*
t	*Time-averaged component of pressure, N/m2*
T	*Wall Temperature,* ^oC
T_f	*Fluid` Temperature,* ^oC
Re	*Reynolds number*
V_a	*Axial velocity m/s*

V_r Radial velocity m/s

V_t Tangential velocity m/s

ρ Density, kg/m3

μ Dynamic viscosity, kg/ms

w Width of blade, m

t Thickness of blade, m

P_i Impeller power required

R Radial distance of vessel

CHAPTER 1

INTRODUTION

In most of the industries like general chemical, fine chemical, biochemical, biotech, pharmaceutical, cosmetic, personal care, petrochemical, paint industries, food industries, sewerage etc. mixing is a common phenomenon. For the mixing operations mostly agitated vessels are used to determine the degree of mixing, behavior and performance of the mixing tanks are influences the product quality (Barbara et al., 2005). It is also important process economics point of views. The studies also influence the heat transfer investigations due to the establishment of new methodology of design and assessing the agitated vessel performance. Thus, it is desired to study the temperature which should to be maintained in agitated vessel for well mixing (Reddy et al.2012). The rate of heat transfer to / from the stirred vessel liquid is a function of physical properties of the fluid, vessel geometry, position of agitator and the degree of agitation. The several numerically and experimentally studies have been done for such type of phenomenon (Montante et al. 2001, Armenante et al.1997). A few studies have been reported either experimentally or numerically (Raguraman et al. 2012, Dostal et al. 2010, Karcz et al. 2002). Heat transfer in stirred vessels is important because fluid temperature in the agitator is one of the most significant factors for controlling the outcome of process.

The study of the heating process in stirred tanks is of particular importance in the food industry. The object of such a process is to uniformly raise the temperature of the vessel contents, thereby allowing a desired process to occur. The quality of the heat distribution in a particular system is dependent on many factors, such as impeller type, vessel geometry configurations, manner and rate of heat input, impeller rotation speed, properties of the mixture and so on.

The numerical simulations have been developed rapidly in recent decades, the CFD method has became a useful and powerful tool (Bakker A., 1996). Numerical modeling of heat transfer with the use of Computational Fluid Flow (CFD) has been widely applied in process engineering. In return, the numerical simulation based on CFD tools may provide good knowledge of flow field and heat transfer information inside the vessels. It reduces the need for extensive experiments and techniques (Shan et al., 2008). The numerous programs have been written to solve either specific problems specific classes of problems. From the mid 1940's-1970's, the complex mathematics required to generalize the algorithms began to be understood general purpose CFD were developed (Bakker A., 2002). These began to appear in the early 1980's and required what were then very powerful computers, as well as an in depth knowledge of fluid dynamics, and large amounts of time to set up simulations. High speed (performance) computers have been used to solve fluid flow problems for many years. The accuracy of obtained results depends on grid quality (fine grid) and high quality grid is required high performance computers. Hence, it requires a trade off in between grid quality and available computers resources.

Consequently, CFD is a tool used almost exclusively in research. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labor intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost effective and accurate alternative to scale model testing with variations on the simulation being performed quickly offering obvious advantages.

The methodology of the CFD simulations of the turbulent flow and heat transfer is to use numerical methods solving the time-averaging equations describing the conservation of mass, momentum and heat in fluids, together with some approximations. The whole physical space of interest is divided into a large number of small cells (known as 'the grid'). The generation of the grid is the first and the most important step of setting up a CFD simulation. The number and distribution of grid cells can affect the accuracy of the simulation, the solution convergence speed, or even whether the right solution is obtained or not. With the aid of fluid property information and the boundary conditions, the turbulence model equations are numerically solved for each cell in the grid. The calculation is carried out by iteration. That is, the values of all the variables (three components of the velocity, pressure, temperature and related scalars etc.) are estimated at the beginning of the iteration. These values are then updated by feeding them into the corresponding equations which the user is trying to solve. If the differences between the updated values and the previous values are less than the desired tolerances the solution is said to have ‘converged’.

1.1 MIXING OPERATIONS There are different types of mixing operations some of them are given below:

- ***Homogenization:*** It is described as the equalization of concentration & temperature differences, which is the most important and the most frequently carried out blending operation (Kerim Y., 2003).

- ***Heat transfer between a liquid & heat transfer surface:*** Blending reduces the thickness of the fluid boundary layer, hence the thermal resistance on the heat transfer & convective motion of the tank content ensure that the temperature distribution within the tank are reduced (Kerim Y., 2003).
- ***Suspension of solid-liquid:*** In continuous homogeneous distribution of the solid partial in the bulk of the liquid by mixing the suspension, settling of the particles as gravity is prevented (Kerim Y., 2003).
- ***Dispersion of two immiscible liquids:*** Dispersion in liquid to liquid systems is associated with the interface area between two immiscible liquids. This achieved by the keep impeller speed low at which one phase is completely mixed in another (Kerim Y., 2003).
- ***Dispersion of a gas in a liquid:*** This operation is to increase the interfacial area between the gas phase & liquid phase. Increasing interfacial area is obtained by gas sparging by stirrers (Kerim Y., 2003).

1.2 MIXING EQUIPMENT

The classification of mixing equipment is made on both predominant flow pattern that it produces and liquid viscosity, which affects the flow created by rotating agitator. Low viscosity liquids show little resistance to flow and therefore require relatively small amounts of energy per unit volume for a condition of mixing to occur. A typical stirred vessel consists of three parts: tank, baffles and impeller. Figure 1.1 shows the typical tank geometry which is widely used in chemical industry. Tanks used in stirrer equipment can be in different shapes depending on the application. These are cylindrical vessel with a flat bottom, round bottom and rectangular vessels as shown Figure 1.2 Flat bottom tanks are used mainly for solid-liquid agitation while the flat bottom tanks suit better for viscous fluids. Impeller is important parts of the stirred tanks which improve the mixing efficiency and suppress the vortex formation. However they increase the power requirements in the mixing tank. Several baffle arrangements are available according to their using purposes. For example they can be fixed on the tank wall or can be set away from the wall (Kerim Y., 2003).

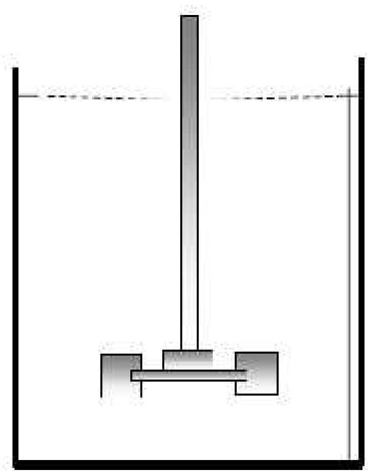
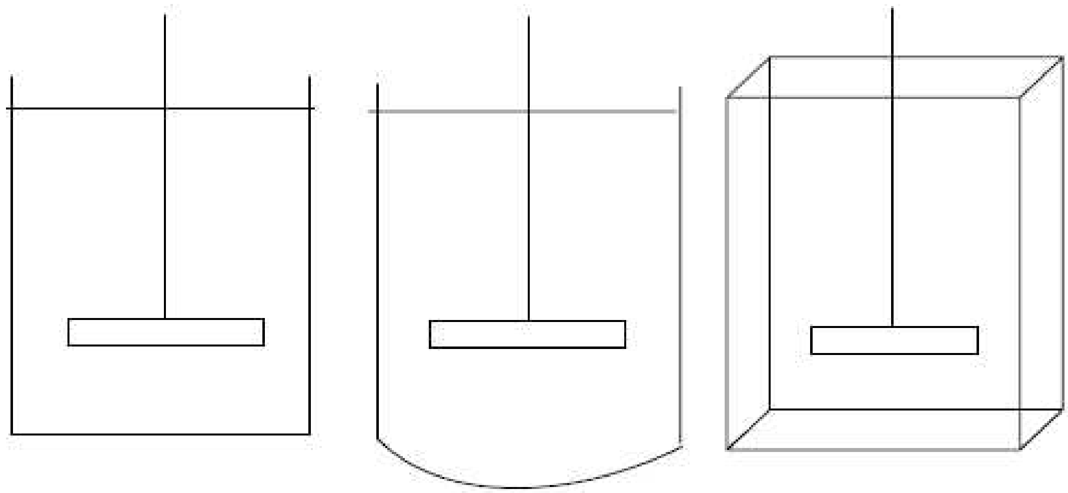


Figure 1.1 Typical Stirred Tank Equipment



(a) (b) (c)

Figure 1.2 Stirred tanks (a) Cylindrical with flat bottom (b) Cylindrical with round bottom (c) Square bottom

CHAPTER 2

LITERATURE REVIEW

The studies of laminar flow regime in stirred vessel can be classified according to the number of impeller blade, concentrically impeller and geometry of the mixing tank, the character of the approaches flow field as laminar and whether it of centric type impeller and tank. The Reynolds number (Re) and Prandtl number (Pr) in this studies, the principle objective have to obtained the flow field variables like velocity, pressure, temperature and derived quantities which allow the determination of engineering parameters like heat transfer coefficient (h), Nusselt number (Nu). The available literature has been reviewed and related studies have been summarized in proceeding sections.

Rahmani et al., 2013; have been predicted an increase in Hedstrom Number (He) with decrease in velocity. A little bit change in the yield stress implies a change in the behavior of fluid flow. Mainly contribute focuses on the viscoplastic fluid, which is not less controlled, particularly the polymerization in emulsion. This type of system does not work if we consider two substance of this type of fluid. The field speeds are analyzed and note the existence of a threshold of flow characterized Hedstrom number can lead to a quasi-immobilization of zones inside the system of agitation.

Ashok et al., 2012; have been proposed a mathematical model to analyze the heating rates and that model has compared with the experimental results. Low shear rate concentration of sodium carboxymethal cellulose fluids with two different coil lengths 2.362m and 2.82m, diameter of the helical coil equal to 156mm. The internal & external diameter 4.0mm & 6.4mm were used to correlate overall heat transfer coefficients in an agitated vessel with four blade paddle impeller. The proposed model was derived by using velocity and energy field obtained for straight tube and later extended to helical coil. The new design relation for obtaining the individual heat transfer coefficient in terms of flow behavior index is equal to

Raguraman et al., 2010; had worked on heat transfer performance in stirred vessel in coal water slurry system using coal gasification. The effect of the geometrical parameter of blades on heat transfer coefficient was experimentally studied for agitated vessels using coal slurry in coal gasification. The intensity of hear transfer during mixing of fluids depends on the type of the stirrer, the design of the vessel and conditions of the process. The type and size of the stirrer, as well as its location in the vessel, also affect the rate of hear transfer. Besides, the Taguchi method can successfully be applied to heat transfer investigation to save energy, time and material in experimentation.

Babu J., 2010; had investigated on the heat transfer coefficient in an cylindrical stirred vessel using immiscible liquid systems was made at different operating conditions. A spiral coil having material of copper was used as for the provision of cooling water system. The effect of heat transfer on the fundamental parameter like speed of stirred, liquid properties and position of agitator were analyzed and studied the empirical correlation was proposed for heat transfer coefficient of Nusselt number combines with Reynolds number, Prandtl number, volume ration and depth of turbine.

Driss et al., 2010; have predicted the flow field in a stirred tank with the pitched blade turbines and to choose the most effective agitation system. The computer method permits the analyses of turbines with complex geometry and the list of nodes belonging to the interface separating the shape and the flow domain, the meshes generated on a finite volume discretization. The 3D dimensional flow of a fluid is numerically analyzed using the Navier-Stokes equations and the effects of inclined angle on the local flow characteristics have been particularly determined. To verify the simulations the power numbers were measured and compared with computerized results. The flow patterns predicted have been compared with experimental results.

Shan et al., 2008; had investigated CFD simulation provides method for flow field in complex fluid in mechanically stirred vessels. The attention was given to stirred tanks without baffle with axial impeller for frequently used in process industries. The study was intended to evaluate the numerical predictions the properties related to the mixing against measurements. Geometry of three blade Pitched turbine down flow was used to generate axial flow field in a s tank. Eulerian reference frame was proposed for the impeller and the bottom of the tank. It predicted results of two phase flows at different impeller speeds; various laminar viscosity coefficients were tested to improve the predictions. The particle distribution was affected with the impeller speeds in the range of 113-173 rpm in the agitated vessel. The concentration of solid particle reported different profiles at different axial sections. Good agreement between the experiments and simulations was observed. The steady and unsteady laminar mixing processes examined in stirred tank with a plate impeller. Lagrangian particle method based Semi implicit MPS technique adopting. It is observed that the mixing is strongly dependent on the initial configuration of the mixing interface of fluids. In steady mixing there found to be side by side fluid configuration, while unsteady was better annular fluid configuration, radial mixing for those some fluids residing near the vessel wall. It observed that there was certain unsteadiness on the rotational velocity of the impeller tip, which may serve as alternative in replacing baffles to promoting radial flows near the stirred tank.

Barbara et al., 2005; had worked on **r**esults shows that when compared the experiment value of local heat transfer coefficient, h, with CFD modeling of turbulent heat transfer in stirred tank using k-e model the difference were equals 12.2%. The equations used in this work were Navier Stokes Equation, Continuity Equation, Energy Transport Equation having parameters, Vessel Diameter (T)= 0.158m, Blade Diameter (D)= 0.483T, H=T. The value of heat transfer coefficient was not known to free surface of stirred liquid & bottom of the tank in all tested cases. The results were good when compared experimental and CFD modeling results.

Torre et al., 2006; had investigated the power which decreases with increase in frequency. The current work with k-e model has limitation when used in transient mode. In future work Scale Adaptive Simulation (SAS) model, for the model to be useful in design mixing systems convergence times and computer requirements must remain practical. The work investigated that there was significant transient effects that mean many of the “rules of thumb” that have been developed for fully baffled vessel must be revisited and a complex, transient flow structures generated by the impeller interactions.

Kumaresan et al., 2005; the flow pattern for power consumption and mixing timing in a agitated vessel depend not only on the impeller design but also on the geometry of the stirred tank . For the measurements of power consumption, flow field and mixing timing pattern had been carried out in a stirred vessel with 0.5 m diameter and standard 45° pitched blade turbine hydrofoil impeller with a variety of baffle configurations. The comparison of the flow pattern average velocity, turbulent kinetic energy, energy dissipation rate, average shear rate had been presented on the basis of equal power consumption. Comparisons of interaction between the impeller and the internal flow with laser Doppler anemometer (LDA) and computational fluid dynamics (CFD) predictions have been presented.

Engeskaug et al., 2005; had been done experimental studied for local wall heat-transfer coefficients for water and for the non-Newtonian systems consisting of different percentage solutions of arboxymethylcellulose in water. The results were compared to existing correlation and good agreement was found for system outside the impeller region. For the given systems, a functional dependence between the Reynolds number (Re) and the heat transfer coefficient, given by proposed Nu with Re 0.8 was found to fit the data. This is in contrast to the currently accepted on a Reynolds number exponent of 0.67-0.68. On the basis of the data, a new correlation for non Newtonian fluids has been suggested. The temperature for radial profiles for the wall region was also presented.

Mingzhong et al., 2004; had been predicted the comparison of experimental and numerical simulation of the flow in a vessel stirred by a retreat curve blade impeller are reported. The numerical simulation was carried out using CFX code 5.5.1. The numerical results have been validated through the phase resolved with averaged LDA measurements to avoid the errors associated with pseudo fluids. It was shown that the axial, radial velocities were well predicted over the whole vessel by the numerical simulation, but the tangential and turbulent kinetic energy were not much close in agreement with the experimental data.

Mununga et al., 2003; had been purposed numerical simulations of flow in an unbaffled mixing tank agitated by a plain disk. The circulations and discharge flow rates and their respective non-dimensional numbers were studied. The circulation flow rate was observed to be dependent on the *Re* and also on the size of disk. The bottom clearance range $0.25 \leq C \leq 0.50$, has been seen to produce the discharge and circulation flow and smaller clearance $C = 0.15T$ resulted in reduction of circulation flow rate, circulation flow number and discharge flow number. The discharge flow rate was seen to be dependent on both the size and Reynolds number. The values of circulation flow number and discharge flow number found here are smaller than for radial flow impellers. Both the circulation flow number and discharge flow numbers appear not to be constant over range of Re.

Khopkar et al., 2003; this work axial flow with pitched blade impeller was studied, for gas liquid system in stirred vessels. The experimental and numerical study had been focused, models to investigate gas liquid flow generated by the impeller. In PIV measurements fully baffled stirred vessel with diameter 0.19m dished bottom. The measurements of the flow field with and without gas dispersion were carried out. This study made to capture key details of the trailing vortex, accumulation of gas and the flow around the impeller blades. For the simulation of impeller rotation CFD code, FLUENT 4.5, approach was used. The predictions were find out by comparison with the PIV and other available experimental data. The results of CFD model were useful for better understanding and simulation of flow field generated by axial impellers in stirred vessels.

Stephens et al., 2002; had been calculated experimental heat transfer data is presented for two batch operations of flow mixing. In one case fluid is oscillated within a baffled tube and in the second case baffles are oscillated within a process fluid. Both situations the heat transfer coefficient depends on the intensity of oscillation, and the energy of each configuration corresponds to that of an equivalent net turbulent flow in a pipe or a batch stirred vessel. The results indicated that flow batch mixing is as energy efficient as other conventional mixing configurations and the heat transfer performance indicates that each oscillatory flow mixing configuration could be satisfactorily used as a batch reactor system.

Shekhar et al., 2002; had been predicted CFD simulation of the flow field, power consumption for a mechanically stirred eight blade paddle impeller in an unbaffled vessel geometry over a range of Reynolds numbers covering laminar, transitional and turbulent regimes. The calculations were performed using the sliding mesh technique to account the impeller motion power consumption were done using a simulated tracer injection experiment. The grid density and the choice of the model were investigated and the results are compared with data from Dong et al. There was good agreement find out and show satisfactory results. The rotational speed of impeller remains constant for laminar flow and prediction is good for mixing time.

Karcz et al., 2002; had performed an experimental study on the efficiency of the heat transfer process in a jacketed agitated vessel of different slenderness, equipped with eccentrically located HE (high efficiency) impeller or propeller. The efficiency of the process was evaluated on the basis of the measured values of the heat transfer coefficient and power consumption. The measurements were carried out in the turbulent flow of a Newtonian liquid. Liquid aspect ratio of the agitated vessel was varied from 0.5 - 1.5 and the eccentricity of the turbine shaft was in the range from 0 -0.53. The measured quantities were approximated analytically.

Jaworski et al., 2001; had investigated a combined experimental and numerical study for flow associated with six bladed pitched blade turbine at three different bottom clearances. Good quantitative agreement for flow field had been obtained but that for RMS value was poor. The mean discharge angle is more vertical for the smaller impeller at all clearances and for both sizes when an impeller approached a surface towards which it is pumping.

Mununga et al., 2001; had been summarized laminar flow study in an unbaffled stirred vessel stirred by a disk and a 4-blade impeller was simulated using the CFD code FLUENT. The geometries and grids were generated using a pre-processor software package (Gambit) and then exported to Fluent where the rotating reference frame approach was employed for the flow field. The flow field at center is characterized by two recirculation regions. The swirl velocity and the vorticity at centre are much higher for the 4-blade impeller than for the solid disk. The numerical results also show that both the velocity and the vorticity are higher near the impeller tip. The results at Reynolds number ($20<Re<200$) the 4 blade impeller there is increase in the swirl velocity near the axis at the bottom of the agitated vessel. The results show that the disk impeller rotor be viable for smooth laminar flow field applications.

Montante et al., 2001; had been carried study for distribution of solid particle with baffled stirred vessel with four Pitched blade turbines (PBT) was investigated using both experimental measurements and CFD simulations. Diluted suspensions of glass bead in water and moderately viscous liquid were considered. The measurement of axial velocity profiles was conducted by light attenuation technique. System was performed using a Sliding Grid approach and homogeneous two phase k-e turbulence model. The experimental and numerical simulated particle axial concentration profiles were compared and good results were found.

Delaplace et al., 2001; In this work, heat transfer had been investigated for several Newtonian and non-Newtonian liquids agitated in a rounded bottom vessel equipped with an atypical helical ribbon impeller. Both experimental and numerical approaches were attempted. The measured rates of heat transfer from the wall of the vessel to the agitated liquid allow the comparison of heat transfer performances of this mixing system with those obtained on standard helical ribbon impeller. A method was indicated to correlate non-Newtonian results with those established with Newtonian liquids. No temperature gradient was detected in the bulk. Numerical approach allows the local heat transfer coefficient to be obtained, and underlines the difficulty to relate global heat transfer coefficient to the thickness of the thermal boundary layer.

Curran et al., 2000; had been investigated batch mixing in the laminar flow regime was studied using single and double helical ribbon impellers and a series of fluids prepared from the water and sodium hydroxide. Power draw measurements for these fluids were compared to the values obtained for comparably high viscosity Newtonian fluids. In the mixer circulation times as a function of impeller speed were measured. The dimensionless circulation times were found to be independent of direction of pumping and fluid rheology. For the highest viscosity fluid the single impeller with downward pumping gave a high value of dimensionless circulation time. For the single impeller, the dimensionless circulation time with upward pumping was less than 2/3 of the previously observed values for Newtonian fluids.

Lamberto et al.,1999; had compared the laminar flow field inside an unbaffled stirred tank generated by a six blade radial flow impeller was characterized using e experimental and numerical simulation. As expected the flow structure in the system were two vortices above and below the mixing impeller. These circulation regime were segregated the flow and for low Reynolds numbers 5< Re <100 and found that their positions and size were depend on Re and on the position of the impeller blades. The pumping and circulation flow of the stirred were quantified and indicate results that the circulation flow was approximately four times the pumping capacity of the impeller.

Bakker et al., 1998; had anticipated that a finer grid is required to resolve the effects of possible turbulent tip vortices. The sliding mesh method can be used to obtain accurate solution to predict the time dependent laminar flow pattern in stirred reactors. T=0.3m, D=T/3, C=T/3.

Armenante et al.,1997; had been studied velocity profile and the turbulent kinetic energy were obtained for the flow generated by a 45 ° pitched 6 blades turbine in an unbaffled tank with flat bottom cylindrical tank and fully filled with water. The mean velocities fluctuations in different directions were experimentally measured LDV technique with height variations and radial positions within the vessel. A computational fluid dynamic (CFD) code, Fluent was used to numerically predict the velocity distribution, fluctuates, power consumption, and pumping capacity of the impeller. The comparison of experimentally and numerically obtained mean velocities, turbulent kinetic energies on the top, bottom horizontal surfaces by the impeller were used as boundary conditions . The validation agreement between the experimental and the numerical predictions was found to be significant. The tangential velocities were found to be significantly bigger than the other two velocity components and strong radially flow was observed. There are two main recirculation flows above and below the impeller and the dimensionless mean velocities components were found to be nearly independent of the impeller speed.

Jaworski et al., 1997; had been presented study for laminar flow with baffled stirred tank. The comparisons of Computational Fluid Dynamics (CFD) package with sliding mesh facility and experimental values for the flow characteristics shows a very good agreement without the need to input any experimental values for the boundary conditions. It was proposed that the model and experiment error ratio adopted as a criterion for the quality of CFD simulations. The obtained ratio in this work was now to performed numerical study for the process engineering aspects and integrated design of agitated vessel, mechanical engineering and other related issues.

Haam S. et al.,1993., it reported that local heat transfer sensor system was utilized to determine heat transfer coefficient associated with mixing in blending vessels with a high efficiency impeller with different diameters was used to determine its effects on the mixing parameters of the tank. A tank 0.38 m with a fluid depth of 0.46 m was determined. The off bottom clearance of the impeller was 6.35 cm. Impeller speed was studied a range of Reynolds numbers. The empirical heat transfer equation was examined for each component.

Warren et al., 1965; had worked on velocity profiles measured in a flat bottomed agitated vessel with baffles in the flow regime close to the wall. The velocity profiles near the wall surface were predicted as functions of impeller type, size and speed for varying vertical wall locations. The velocity profile (experimental) data were used with the results of a prior heat transfer study to illustrate that the transfer of heat from the wall of an agitated vessel. The magnitude for the transfer in flat plate boundary layer flow calculated.

CHAPTER 3

PROBLEM DESCRIPTION, MATHEMATICAL MODELING & GOVERNING EQUATIONS

The detailed problem description along with simplified assumptions is presented in this chapter. The simplified assumptions reduce mathematical complexity and provide appropriate governing equations, which are presented in this chapter.

3.1 PROBLEM DESCRIPTION

The system investigated in the present work is the flat cylindrical bottom with unbaffled cylindrical tank equipped with a four Pitched blade turbine (PBT). The tank geometry and impeller boundary conditions are similar to Mununga et al, 2001. The schematic picture of present study system is as shown in Figure 3.1

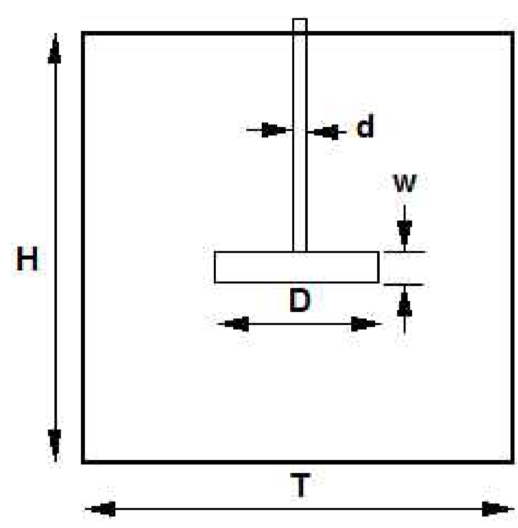


Figure 3.1 Geometry of tank

In the given Figure3.1 T, H, D, w, d, t are the vessel diameter, height, impeller diameter, width of blade, shaft diameter and thickness of the blade. Tank vessel completely filled with fluid having density 'ρ' and viscosity 'μ' which is constant for flow field in laminar regime. The impeller having 4-Pitched blade geometry and the numerical simulation for flow field and heat transfer are done by fixed the impeller and moving the reference frame. The dimensional ratio for whole geometry of stirred vessel as shown in Table 3.1

Table 3.1 Dimensional Ratio for Agitated Vessel

Item	**Symbol**	**Ratio**
Vessel Diameter	T	T
Height of Vessel	H	$H=T$
Impeller Diameter	D	$D=0.4T$
Bottom Clearance	C	$C=0.5T$
Impeller Shaft	d	$d=0.025D$
Width of Blade	w	$w=0.2D$
Thickness of Blade	t	$t=0.05D$

3.2 FLUID PROPERTIES

Simulations are predicted for laminar regime corresponding to Reynolds number (Re=20). The Re used in this study was based on the impeller diameter as shown in equation (3.7). Density and viscosity is constant flow analysis and fluid having density 1.0 kg/m^3 and viscosity 0.1 kg/ms.

3.3 BOUNDARY CONDITIONS

For the steady state condition and to simplify the boundary conditions, a rotating reference frame system selected by fixed the impeller and a rotate the tank at same velocity but in opposite direction. There is no slip boundary condition applied on the vessel walls and impeller. A constant wall temperature 330^{o}C has been applied to the bottom and walls of the vessel. A rotating reference frame of the impeller is employed for numerical simulations reported in this chapter. In this reference frame the impeller remains stationary whereas the tank wall is rotating at the same impeller angular velocity but in the opposite direction. Since the velocities are relative to the reference frame rotating with the impeller, the boundary and the initial conditions have specified in term of this rotating reference frame. On solid vessel wall, the conventional linear logarithmic wall function is used; here U_x, U_y, and U_w are the radial, tangential and axial velocities, respectively, and T_r being the radial distance from the central axis of the cylindrical tank. The surface of the liquid is simplified as a flat plane, Therefore, Table 3.2 here shown all initial boundary condition for tank, impeller and shaft of agitated vessel.

Table 3.2 Numerical Boundary Conditions

Item	Name	Conditions
Impeller Blade and Shaft	Stationary	(No slip)
Tank (Outer vessel, Bottom, Top lid)	Moving	Angular velocity

3.4 MATHEMATICAL MODELLING & GOVERNING EQUATIONS

From literature (Mununga et al., 2001, Armenante et al., 1997) is clear that pitched blade impeller are widely used in agitated vessels, because of simple in geometry and dimensions. Three dimensional laminar flow regime and heat transfer in a mechanically agitated vessel are the subject of this work. A numerical approach has been used to describe and predict the flow field and heat transfer. The mathematical model with motion equations and heat equation was applied using the commercial available package of the Computational Fluid Dynamics (CFD) ANSYS, Fluent. The motion and heat transfer equations associated with laminar model are introduced in this chapter to provide the good understanding of the methodology of laminar flow and heat transfer simulations.

GOVERNING EQUATIONS

The present study is performed for steady state incompressible fluid. The assumptions lead to decoupling of momentum and energy equations. Therefore, applying the conservation laws to the incompressible fluid, the following equations are obtained:

Continuity Equation

$$\frac{\partial U_x}{\partial x} + \frac{\partial U_y}{\partial y} + \frac{\partial U_z}{\partial z} = 0 \quad (\rho = \text{consatant}) \tag{3.1}$$

The flow is steady and incompressible and density nearly constant in the flow field (Bird et al. 2002).

Energy Equation

There are many flow field problems in which we are interested to calculate heat transfer rates surface to fluid (Bird et al. 2002). For this differential form of the heat transfer equation is utilized are as follows

$$\rho C_p \left(U_x \frac{\partial T}{\partial x} + U_y \frac{\partial T}{\partial y} + U_z \frac{\partial T}{\partial z} \right) = k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \tag{3.2}$$

Momentum Equation (Bird et al. 2002)

$$\rho \left(\frac{\partial U_x}{\partial t} + U_x \frac{\partial U_x}{\partial x} + U_y \frac{\partial U_x}{\partial y} + U_z \frac{\partial U_x}{\partial z} \right) = -\frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 U_x}{\partial x^2} + \frac{\partial^2 U_x}{\partial y^2} + \frac{\partial^2 U_x}{\partial z^2} \right) \tag{3.3}$$

$$\rho \left(\frac{\partial U_y}{\partial t} + U_x \frac{\partial U_y}{\partial x} + U_y \frac{\partial U_y}{\partial y} + U_z \frac{\partial U_y}{\partial z} \right) = -\frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 U_y}{\partial x^2} + \frac{\partial^2 U_y}{\partial y^2} + \frac{\partial^2 U_y}{\partial z^2} \right) \tag{3.4}$$

$$\rho \left(\frac{\partial U_z}{\partial t} + U_x \frac{\partial U_z}{\partial x} + U_y \frac{\partial U_z}{\partial y} + U_z \frac{\partial U_z}{\partial z} \right) = -\frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 U_z}{\partial x^2} + \frac{\partial^2 U_z}{\partial y^2} + \frac{\partial^2 U_z}{\partial z^2} \right) \tag{3.5}$$

The variables in the equations are rendered dimensionless using diameter of tank (T), velocity U^*, T/U^* and ρU^{*2} as variable for length, time, velocity and pressure respectively. Therefore, the dimensionless groups appearing in Eq. (3.2) and the Reynolds number is defined as:

$$Re = \frac{\rho N D^2}{\mu} \tag{3.6}$$

Prandtl Number (Pr)

$$Pr = \frac{\mu C_p}{k} \tag{3.7}$$

It is defined as the ratio of momentum diffusivity to the thermal diffusivity (Bird et al. 2002).

Nusselt Number (Nu)

It is the ratio of convective heat flux to conduction heat flux in the fluid boundary layer. Larger the value of **Nu** indicates large convection in the fluid (Bird et al., 2002).

$$Nu = \frac{hT}{k} \tag{3.8}$$

CHAPTER 4

NUMERICAL METHODOLGY & GRID GENERATION

This chapter deals with the adopted numerical methodology, the silent features of the grid regeneration and grid independence.

4.1 NUMERICAL METHODOLGY

We have used ANSYS Workbench for the geometry of the agitated vessel. The 4 pitched blade unbaffled tank simulation has been used for single phase steady state. Fluent tutorial guide has been followed for the simulations setup. The geometry stirred vessel as shown in Figure 4.1.

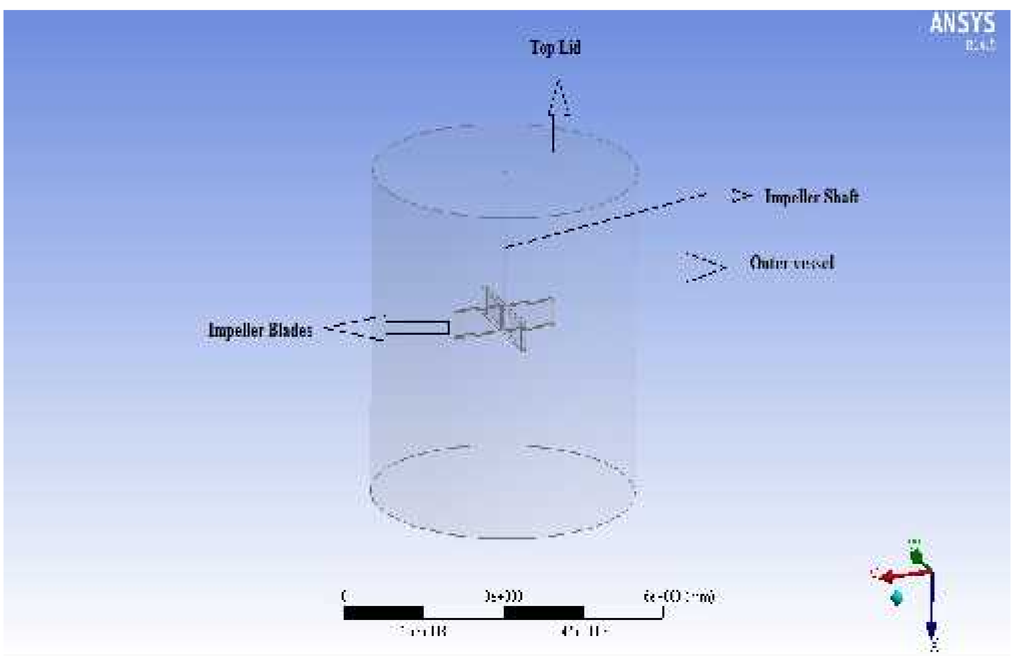


Figure 4.1 Assembly of Agitated Vessel

The Figure 4.1 shows the assembly of agitated vessel having pitched blade impeller generated in ANSY Workbench. The impeller axis is centrally located and the all specifications for the agitated vessel as shown in Table 4.1. The fluid in the vessel is equal to tank height and system is simulated for single phase steady state.

Table 4.1 Specifications of existing model (Mununga et al., 2001)

Item	**Symbol**	**Dimension**
Tank	*T*	*5m*
Height	*H*	*5m*
Bottom Clearance	*C*	*2.5m*
Impeller	*D*	*2m*
Shaft	*D*	*0.050m*
Number of Blades	*N*	*4*
Width of Blades	*W*	*0.4m*
Thickness	*T*	*0.1m*

4.2 MESH GENERATION AND GRID INDEPENDENCE

The shaft and the impeller were stationary in relative velocity formulation of the Rotating reference frame and found that ax symmetrical flow in a closed cylindrical tank with a rotating lid was still stable up to Re=2000 (Gelfgat et al, 1996). The flow observed by other researchers was deceptive. Based on the above findings, the flow generated by the Pitched blade turbine is assumed to be symmetrical in the azimuthally direction and as a result, the four blade impeller is modeled using a 90° sector. Both the geometry and grids is generated using a commercial software package called ANSYS. The tool of meshing in ANSYS Workbench is to provide easy and robust to use meshing tools that is simplify the mesh generation process. These tools in workbench have the benefit of being highly automated along with having a moderate to high degree of user control.

4.3 MESH GENERATION

A tetrahedral meshing is done for the model of the agitated vessel. Course, medium, fine meshing is done. But fine meshing used for the regions were the forces are more significant and the meshing size function is used for all other regions. The sensitivity of the grid tests have also been done for existing model. As far as the analysis is concerned a finer meshing scheme is used to take in consideration of the every vital effect which is used the flow in case of single phase and steady state. The shaded and wire frame mesh tool to be check for mesh generation is as shown in Figure 4.2. For flow structure and heat transfer to agitated vessel, it is projected views either auxiliary or orthographic, which show internal mesh structure of the tank and impeller along the specified cutting plane. These views are commonly used to show internal features with more clarity than may be available using regular projections. In the assembly of agitated vessel the parts in sectional view to check the internal mesh structure of vessel as have shown Figure 4.3.

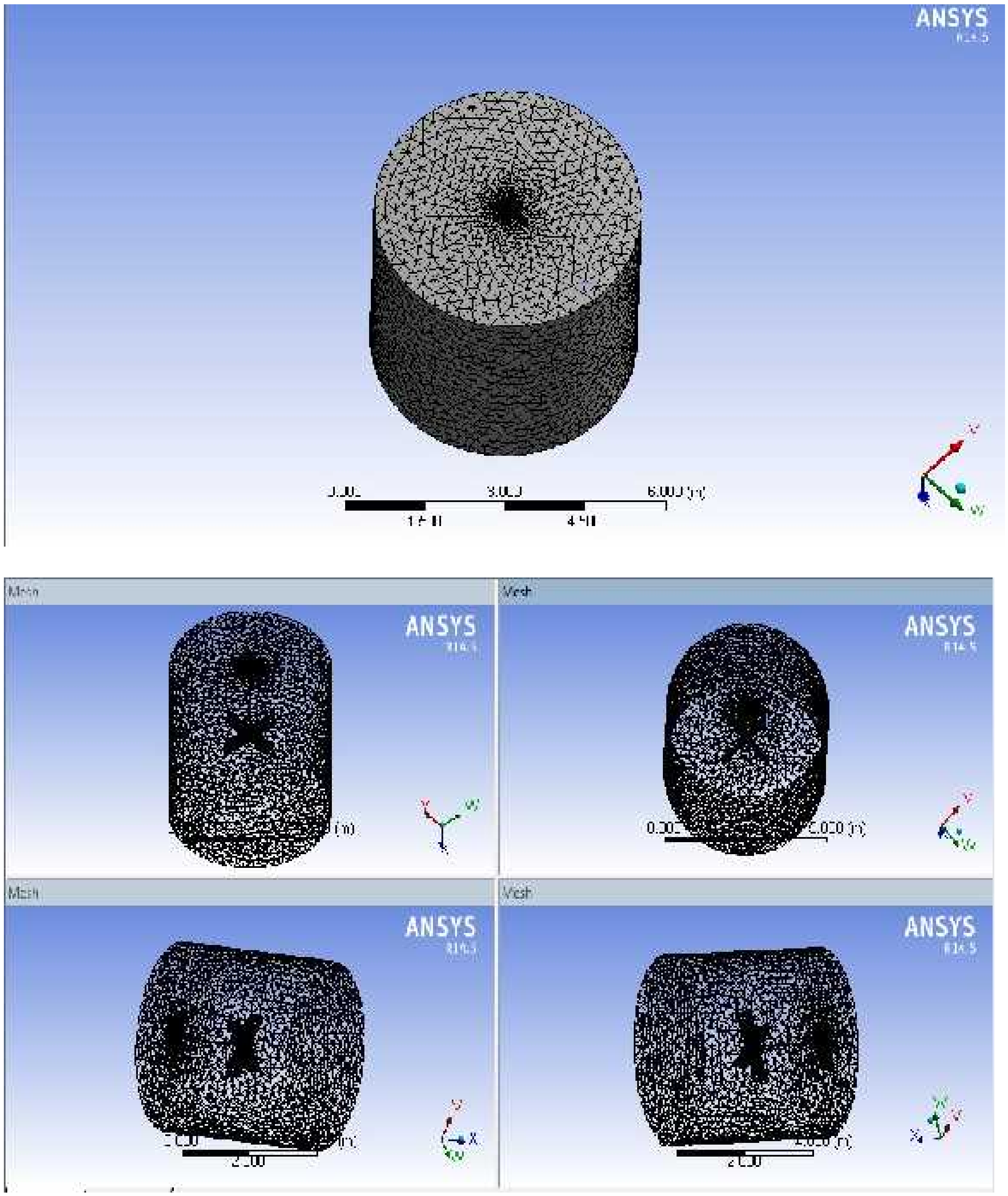


Figure 4.2 Shaded and Wireframe mesh generation in agitated vessel

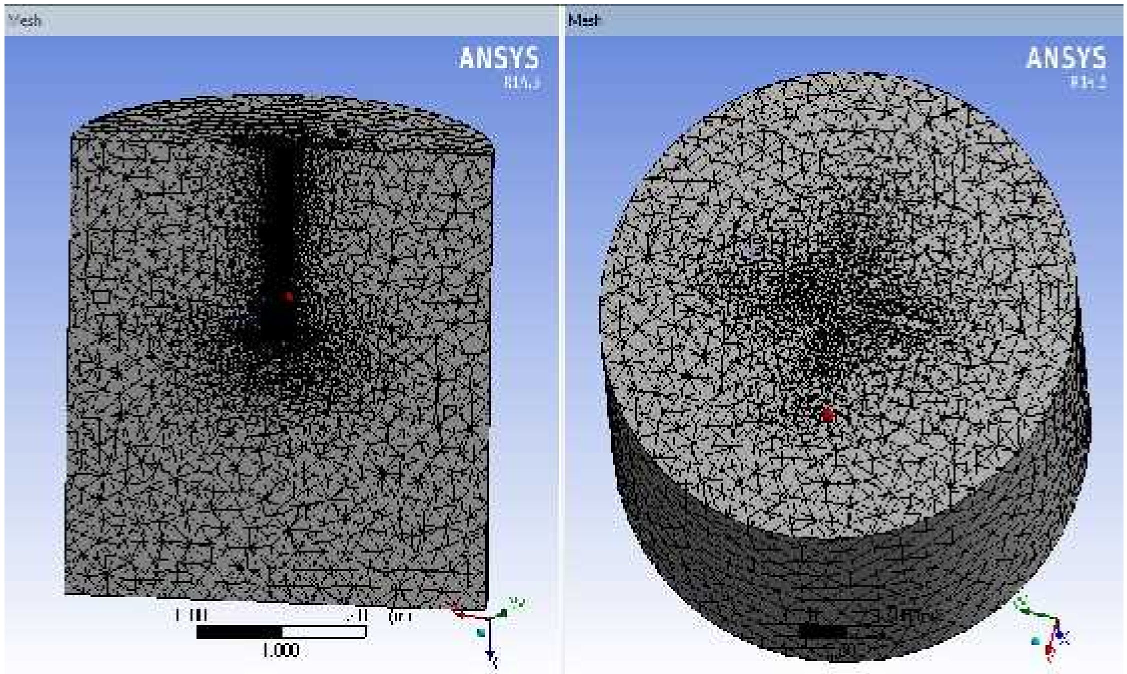


Figure 4.3 Sectional view of the stirred vessel

4.4 GRID INDEPENDENCE

The first step and arguably the most important step in a CFD simulation is generation of a grid (also known as mesh) that defines the cells on which flow variables like velocity, pressure, etc. are calculated throughout the computational domain. Modern CFD codes come with their own grid generator and third party grid generation programs are also available. Many CFD codes can run with either structured or unstructured grids. A structured grid consists of planar cells with four edges 2-D or volumetric cells with six faces (3-D). Although, the cells can be distorted from rectangular, each cell is numbered according to indices (i, j, k) that necessarily correspond to coordinates *x*, *y*, and *z*. Generation of a good grid is often tedious and time consuming engineers who use CFD simulations on a regular basis will agree that grid generation usually takes more of their time than does the CFD solution itself engineer's time, not CPU time. However, time spent generating a good grid is time well spent, since the CFD results will be more reliable and accurate and may converge more rapidly. A high quality grid is critical to an accurate CFD simulation; a poorly resolved or low quality grid may even lead to an incorrect solution. It is important point therefore, for using of CFD to test if their solution is grid independent. The nodes and elements when fine mesh is applied to the given system are having nodes and elements are 80,733 & 4, 59,689. After increasing the mesh size from 4, 59,689 the results are approximately same.

4.5 NUMERICAL PARAMETER

The validation and heat transfer for present study dependent upon most of common parameters used like velocity, pressure, temperature, angular velocity of tank, density of fluid, viscosity of fluid, Prandtl number (Pr) etc. are studied and setup in initial conditions for the flow field and heat transfer in stirred vessel. Here we are used constant speed and temperature for given Reynolds number (Re) and Nusselt number (Nu). The other parameter like capacific heat C_p and thermal conductivity k also used during simulation setup, Table 4.3 shows all parameter related to laminar flow field and heat transfer study.

Table 4.3 Parameters for Numerical Simulation

Item	**Symbol**	**Value**
Angular Speed	*N*	*3.1415 rad/sec*

Item	*Symbol*	*Value*
Density of Fluid	ρ	$1.0\ kg/m^3$
Viscosity of Fluid	μ	$0.1 kg/ms$
Temperature of Fluid	T_f	$300^o\ C$
Temperature of Outer Tank	T_o	330^oC *(Constant)*
Range of Prandtl Number	Pr	*0.71, 7, 20, 30, 40, 50*

The commercial CFD package ANSYS 14.5 is used to perform the numerical simulations. The CFD code uses a finite volume approach to solve the model in hand. For the present study a steady state based solver is used. Here for the analysis of single phase laminar flow regime model is used. The laminar model is selected because of very low Reynolds number. For the pressure velocity coupling we are used SIMPLE algorithm is used which is already mentioned in ANSYS fluent. A moving reference frame condition is used for the rotational domain. Further, different descritization and boundary condition is used for domain. A convergence criterion of for flow validation is $1x10^{-07}$ and for the energy i.e. heat transfer study is $1x10^{-8}$ is taken for all the simulations. All the boundary conditions are summary in the following Table 4.4.

Table 4.4 Summary of Numerical Simulation Setup

Item	Condition
Numerical simulations	3-D
Velocity Formulation	Absolute
Solver	Pressure Based, Steady state

Item	**Condition**
Gradient	Least Square Cell Based
Discritization	Second order
Model for Single Phase	Laminar Model
Momentum	Second Order Upwind
Convergence Criteria	$1x10^{-8}$
Rotating Fluid Zone	Moving Reference Frame

.

CHAPTER 5

RESULTS AND DISCUSSTION

Before moving further it is customarily impeller to check the validity and reliability of numerical process/methodology. Firstly, reproduce the results of Mununga et al., 2001 at Re=20 in Figure 5.1, Good results were predicted error up to 3- 5%

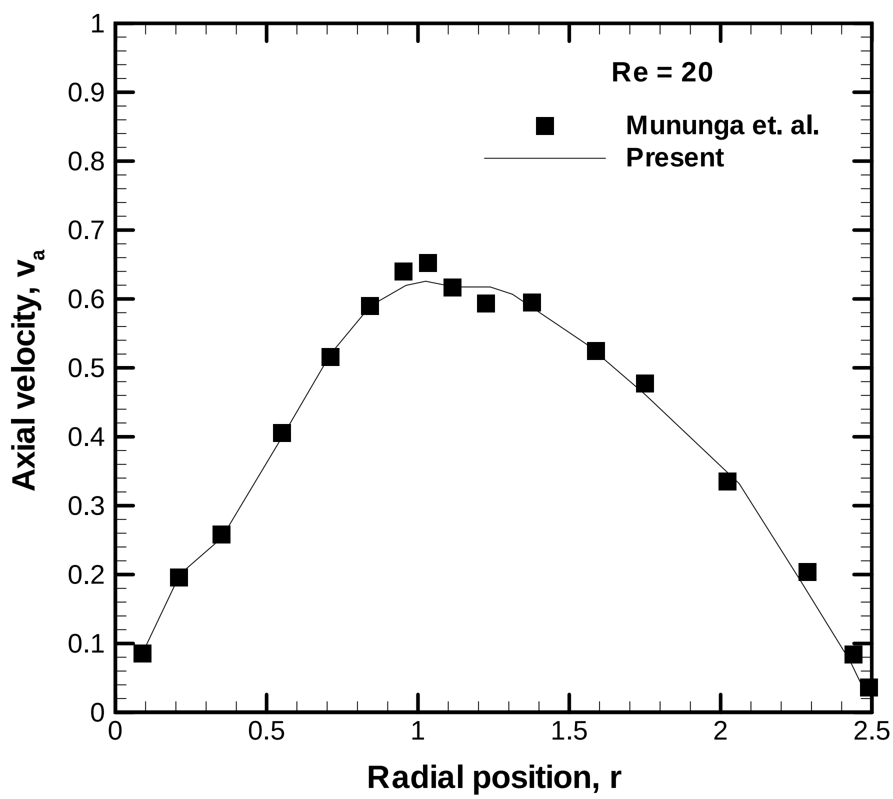


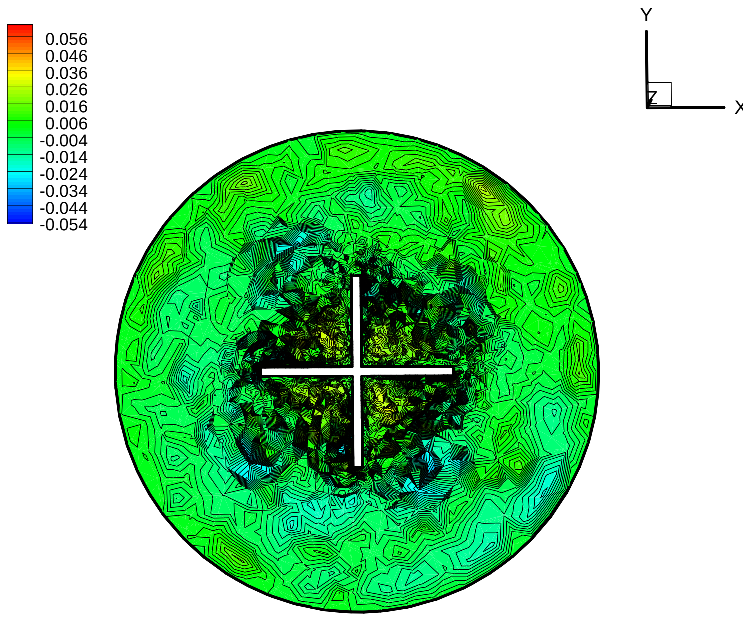
Figure 5.1 Comparisons of present study with numerical data

5.1 FLOW FIELD

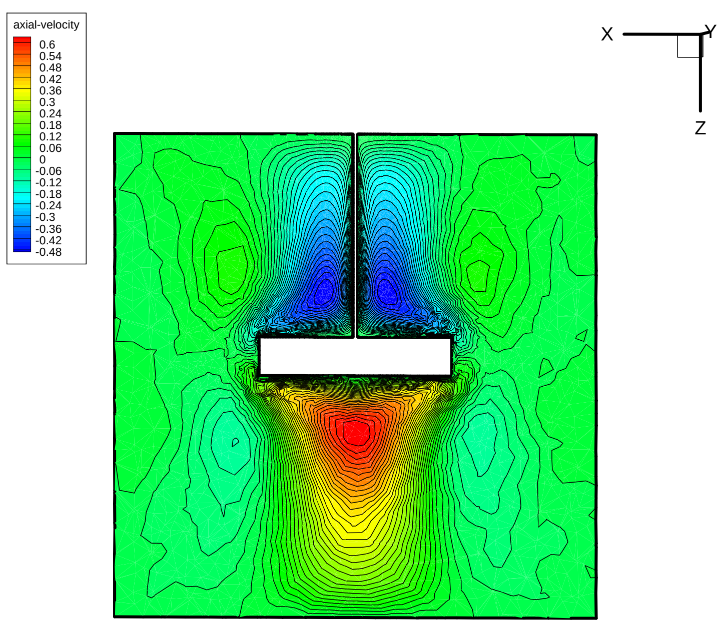
The simulations are performed for a stirred vessel agitated by a four Pitched blade impeller for the rpm 30 in laminar flow regime. Since, the impeller shaft is centrally located; two vertical and horizontal planes velocity profiles for radial, axial and tangential velocities as shown in Figure 5.2, 5.3, and 5.4.

In Figure 5.2, the horizontal and vertical plane shows that the velocity is high near to impeller region and low near to the vessel wall and there is seven recirculation’s loop are observed, four are above the impeller and three are below the impeller region. The predicted results in Figure 5.3 again show nearly same observation radial velocity profile, but the recirculation’s loops are more observed near to vessel wall and impeller region. The contours of tangential velocity profile shows in Figure 5.4 shows stream line variation in region between vessel wall and impeller surface.

Figure 5.5 the velocity profiles on two horizontal planes above and below the impeller, as expected in axial velocity profile more negative loops are observed above the impeller and velocity near the impeller is high. Radial profile shows same velocity profile with more recirculation’s loops.

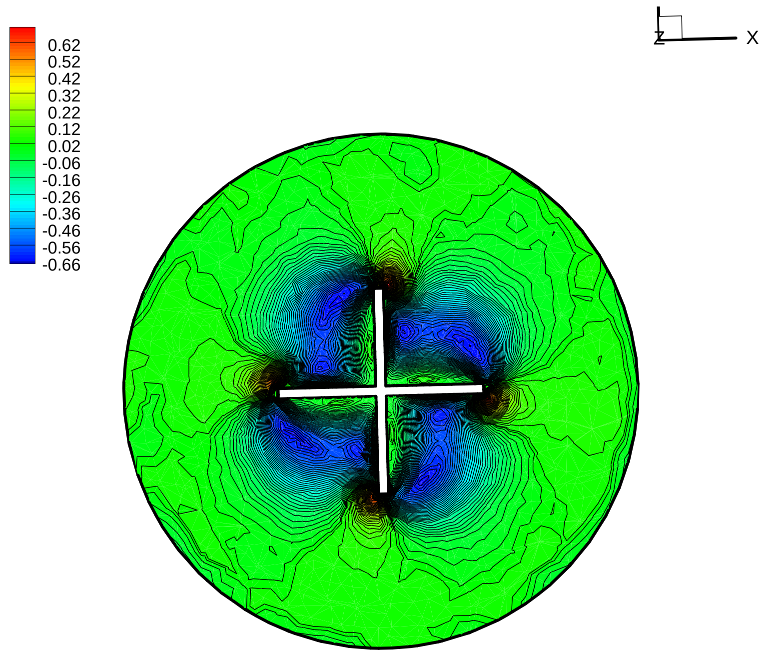


(a)

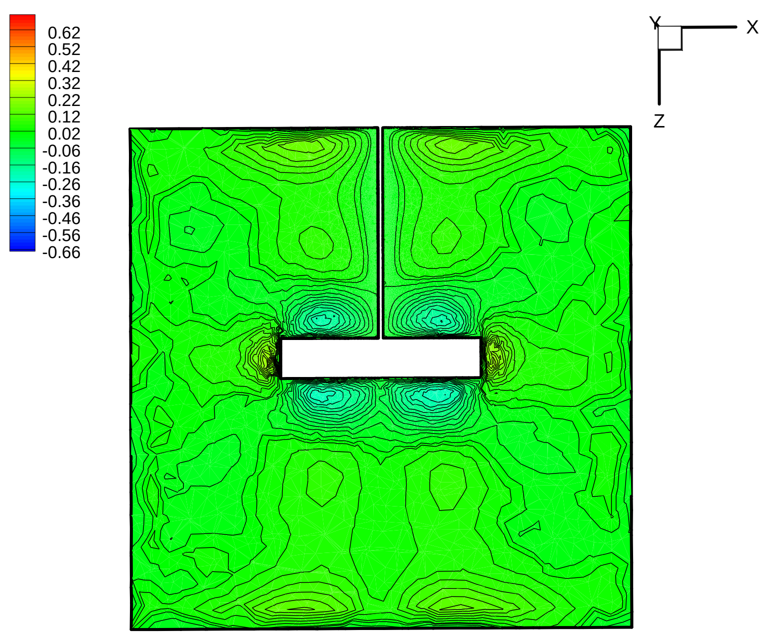


(b)

Figure 5.2 Axial velocity profile (a=HP, b=VP)

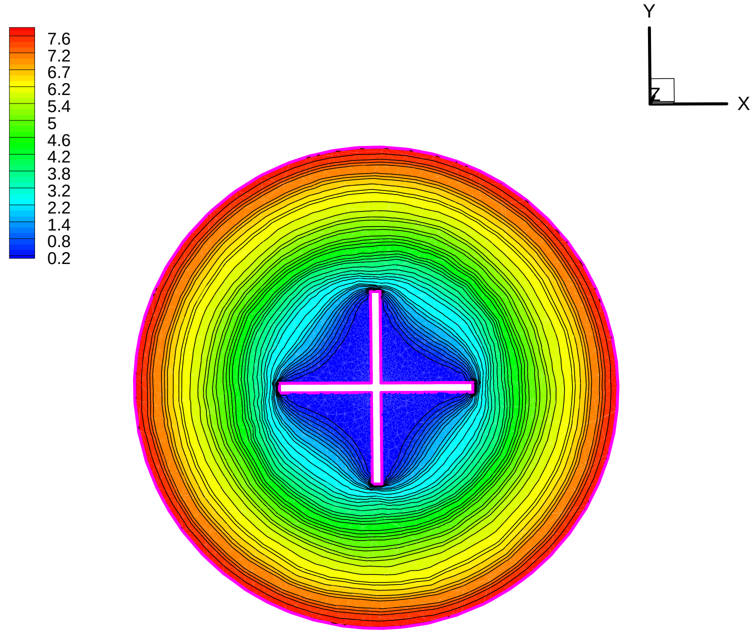


(a)

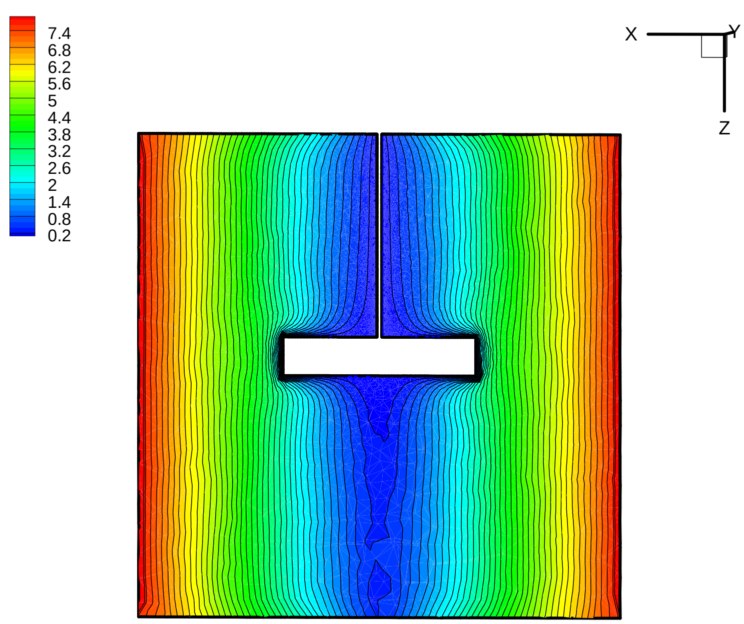


(b)

Figure 5.3 Radial velocity profile (a=HP, b=VP)



(a)



(b)

Figure 5.4 Tangential velocity profile (a=HP, b=VP)

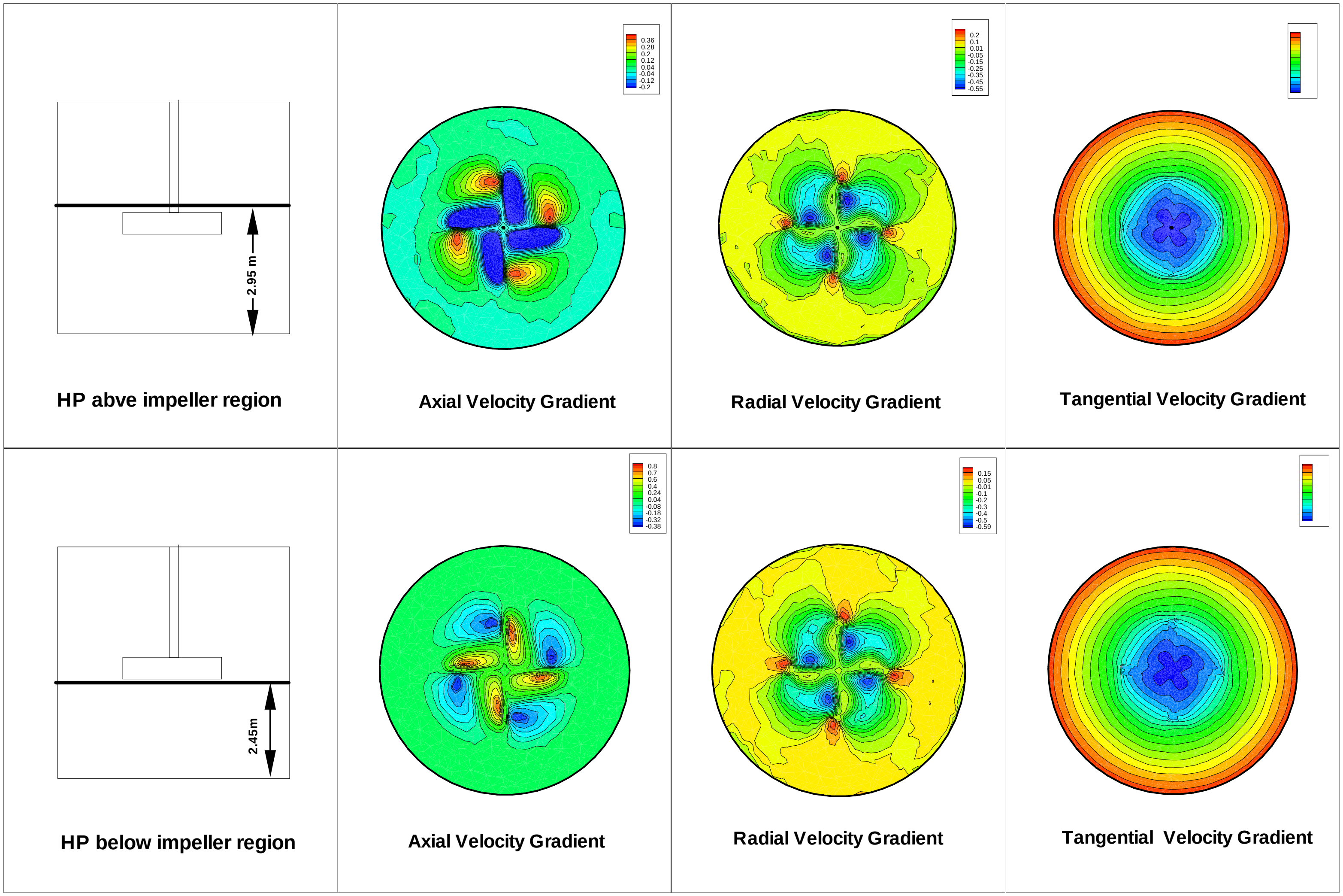


Figure 5.5 Verticals Planes above and below the impeller

5.2 ISOTHERMAL PROFILE

Figures 5.6 illustrate the predicted temperature distributions profile in the steady laminar flow regime for Reynolds number, Re=20. The temperature profile shows that the temperature gradient is having reducing trend. At vessel wall the gradient is very sharp so the isotherms are more clustered near the vessel wall and moderate in the region in between impeller and vessel wall. Near to the impeller the gradient is again sharp due to the agitation process (rotation of the impeller). The maximum heat transfer is found neat the vessel wall because of high temperature difference and it is induced by the circular motion of the fluid due to agitation. These isotherms are parallel to each other and showing circular rings (bands) shown in Figure 5.6-5.8 near to the vessel wall whereas, Hence, in the impeller region isotherms are getting bend to words the axis of the impeller. The vertical section of the isotherm profile is shows that the temperature gradient is again more near to the vessel wall and gradually reducing towards the impeller and shaft and an oval type shape can seen in Figure 5.6-5.8.

In the figure 5.7 and 5.8, the influence of Prandtl number (Pr) is shown on isotherm profiles. The observation of figure 5.7 and 5.8 shows complex influence of Prandtl number on isotherms profile and the isotherms are getting clustered near to the vessel wall with increase in Prandtl number. However, in the region near to the impeller the shape of isotherms become complex with increase in Prandtl number. Overall, a uniform temperature bands are getting thicker with increasing the value of Pr due to the mixing operation. In other words, as the Prandtl number (Pr) increases in the range of 0.71-50, the contours of the temperature profile show better mixing results and low temperature variation.

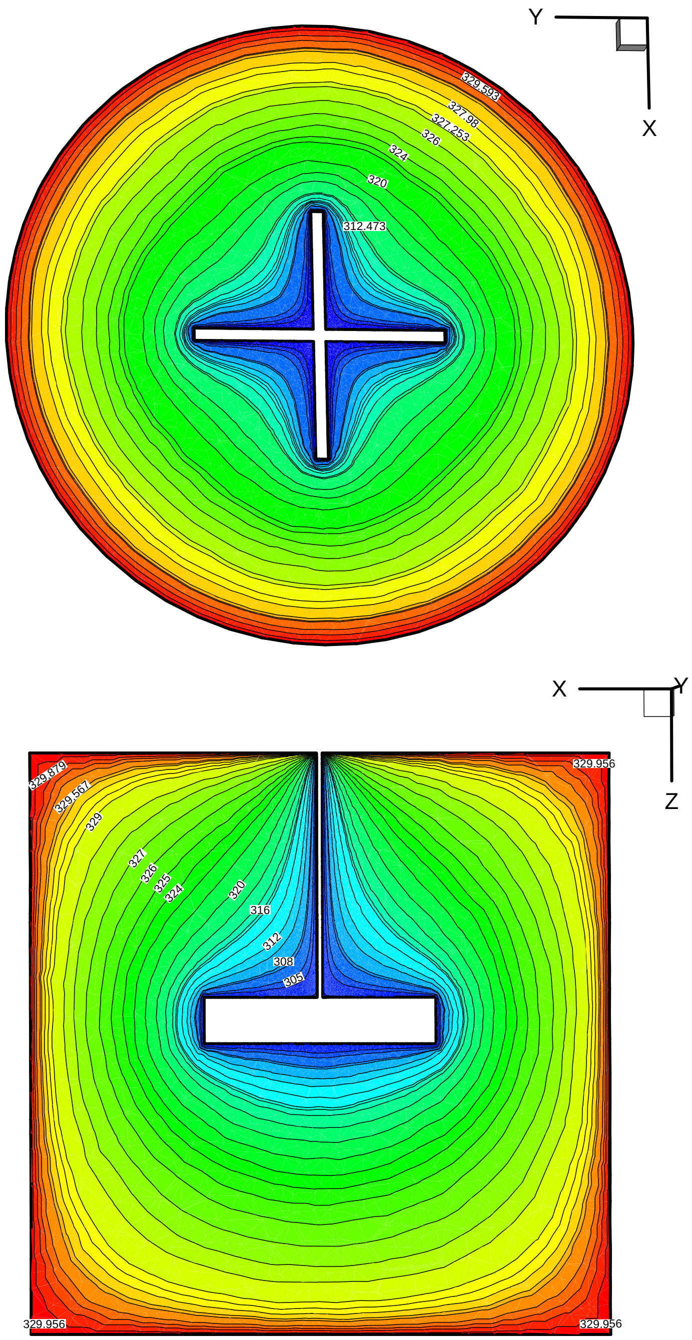


Figure 5.6 Total Temperature Distribution Re=20

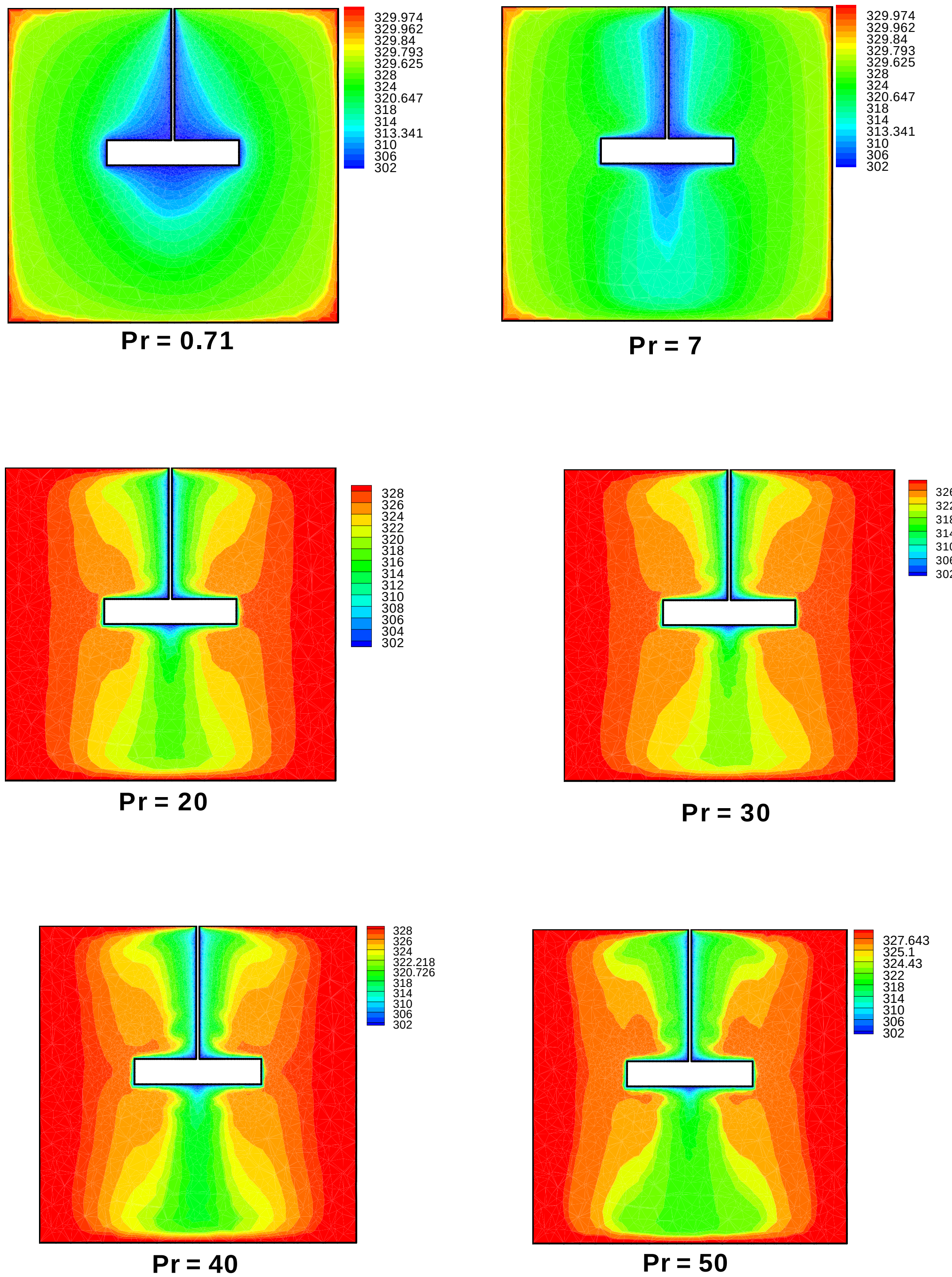


Figure 5.7 Temperature Distribution in VP, Pr=0.71, 7, 20, 30, 40, 50 (Re=20)

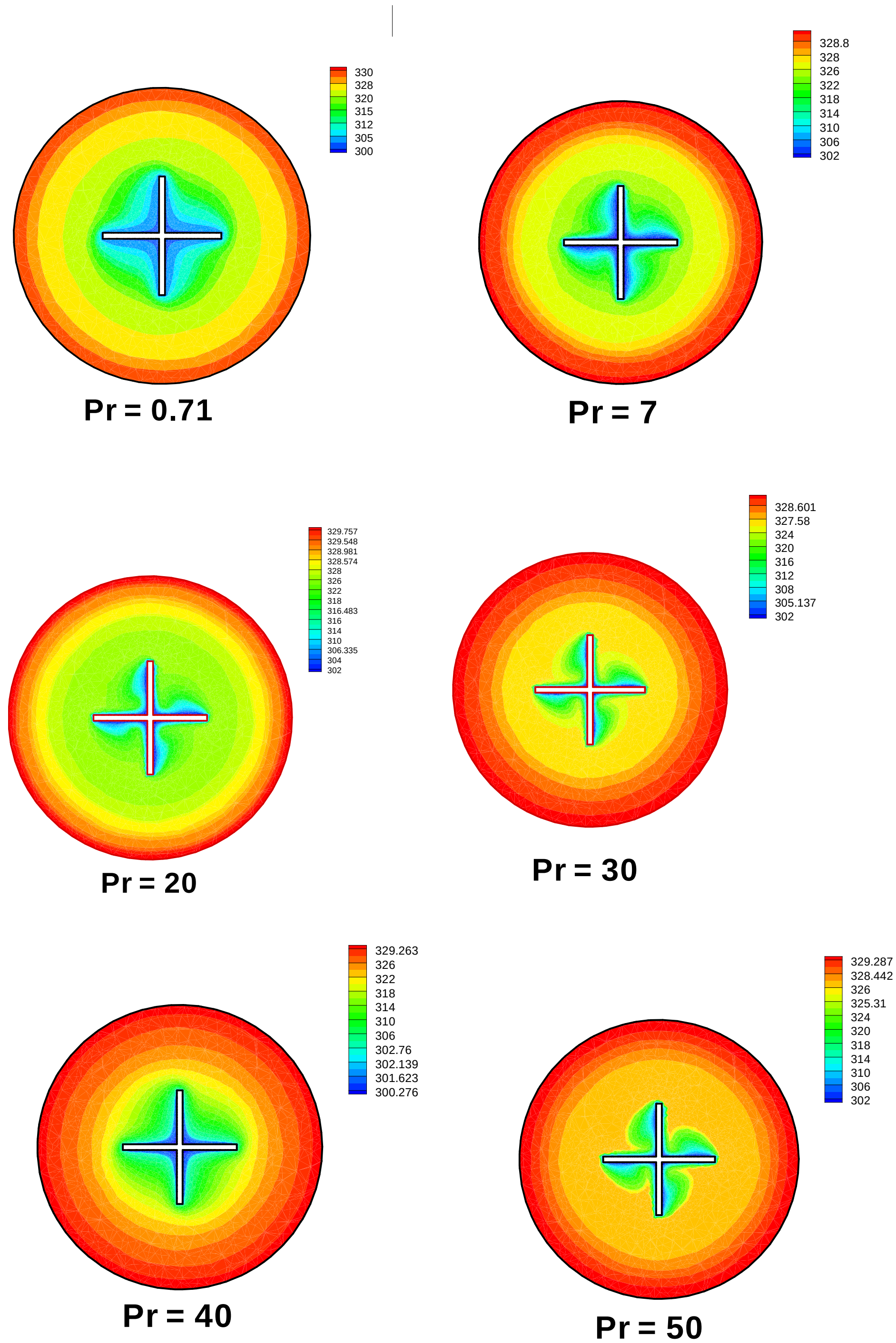


Figure 5.8 Temperature Distribution in HP, Pr=0.71, 7, 20, 30, 40, 50 (Re=20)

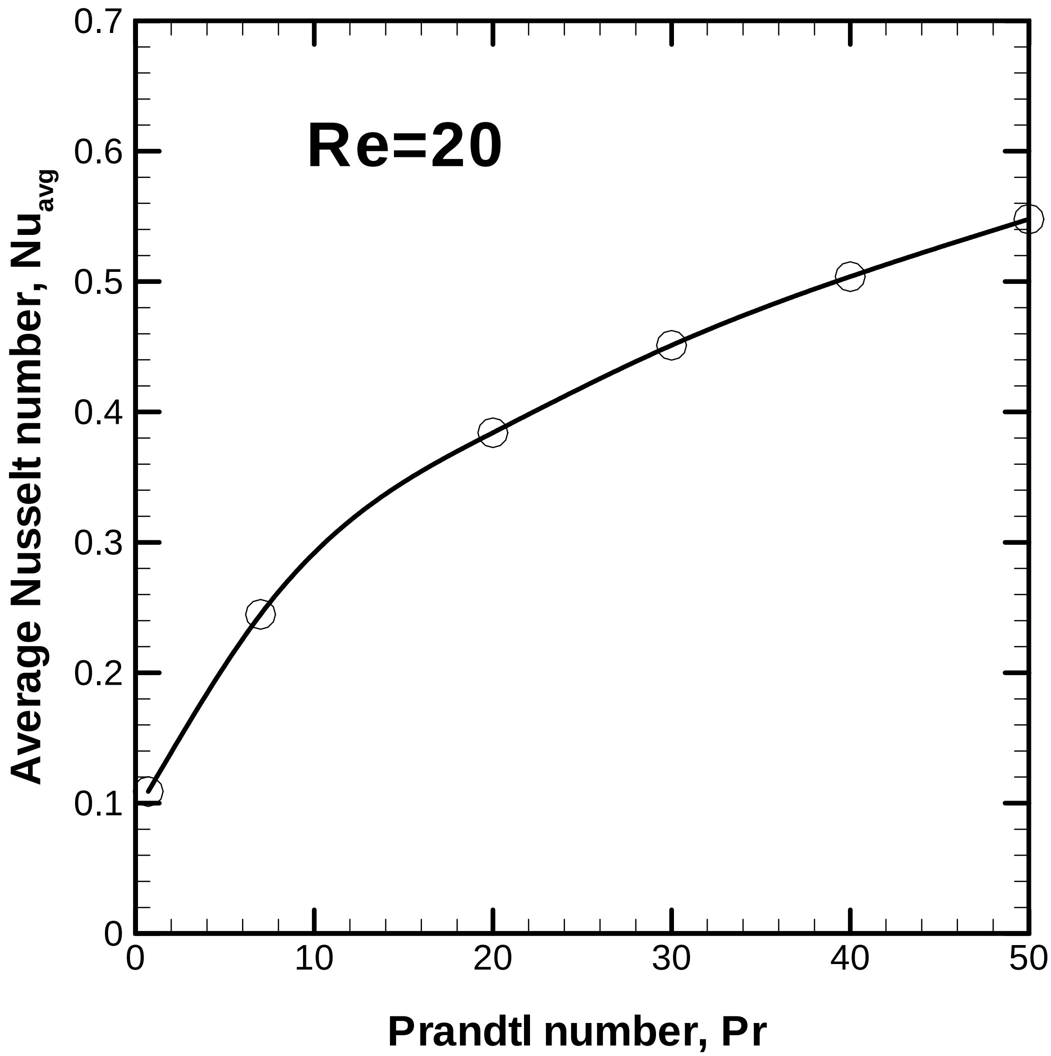


Figure 5.9 Variation of Average Nusselt number (Nu) with Prandtl number (Pr)

Figure 5.9 shows the average Nusselt number (Nu) increasing with increase in Prandtl number (Pr) at this Reynolds number (Re), Similar profile may be seen for another value of Reynolds number in steady laminar flow regime The functional relation for the average Nusselt number and Prandtl number is give by following correlation as

$$Nu=A+BX+CX^2+DX^3 \tag{5.1}$$

Where, X=7.099-7.00, $A=9.23x10^{-2}$, $B=2.33x10^{-2}$, $C=9.5x10^{-5}$, $D=-4.4x10^{-5}$

And present results in the accordance of occurring expected & usual trends.

CHAPTER 6

CONCLUSION AND FUTURE SCOPE

CONCLUSION

The 3-D simulations have been performed to investigate the laminar flow and heat transfer in unbaffled agitated vessel. The validation results have revealed that field produces similar results corresponding to the value for low Reynolds number in laminar flow. Good predicted results are obtained for velocity profile close to impeller region and vessel wall. As simulations performed using rotating reference frame, the velocity is maximum at impeller tip and low near the vessel wall and heat transfer have been in steady state fluid flow in single phase and significant results predicted for Nusselt number (Nu) in the range of Prandtl number (Pr) at constant value for Reynolds number (Re=20) and correlations has been proposed for average Nusselt number as a function of Prandtl Number. Overall, influence of mixing operation increases with increase in Prandtl number.

FUTURE SCOPE

The present research work was aimed for prediction of heat transfer and flow field generated inside the stirred vessel. In order to improve the mixing process which been followed in the present study, there are several that can be augmented.

1. The number of blades of the impeller can be increased for better mixing and heat transfer.

2. Bottom clearance can be considered for further analysis.

3. The impeller angle can be titled and fluid may be Non Newtonian.

4. The geometrical parameters such as impeller diameter, blade width and thickness can be considered for further analysis.

REFERENCES

Aubin J., Fletcher D.F., Xuereb C (2004), Modeling turbulent flow in stirred tanks with CFD: the influence of the modeling approach, turbulence model and numerical scheme, Thermal and Fluid Science, 28, 431-445 .

Armenante P. M., Changgen Luo, Chun-Chiao Chou, Ivan Fort, Jaroslav Medek (1997), Velocity profiles in a closed, unbaflled vessel: Comparison between experimental data and numerical CFD predictions, Chemical Engineering Science, Vol. 52, No. 20, 3483-349.

Bakker A ., Myers, K. J., Ward, R. W., Lee C. K (1996), The laminar and turbulent flow pattern of a pitched blade turbine, Transaction IChemE 74, Part A, 485-491.

Bird R., W.E. Stewart, E.N. Lightfoot (2002), Transport Phenomenon, Second addition, Wiley.

Bakker A. (2004), Stirring up the phases in tall tanks, Fluent Newsletter, 8-9.

Babu J. (2010), Heat transfer studies in agitated vessel using immiscible liquid mixtures, National Journal on Chem Biosis, Vol. 1, 334-341.

Bakker A., LaRoche R.D., Wang M.H., Calabrese R.V. (1997), Sliding mesh simulation of laminar flow in stirred reactors, Trans IChemE, Vol. 75, Part A, 42-44.

Bakker A., LaRoche R.D. (1996), An Overview of Computational fluid mixing applications. Proceedings of the Third World Conference in Applied Computational Fluid Dynamics, Freiburg, Germany, May 19-23, 1996, 3-8.

Driss Z. , G. Bouzgarrou, W. Chtourou, H. Kchaou, M.S. Abid(2003), Computational studies of the pitched blade turbines design effect on the stirred tank flow characteristics, Journal of Mechanics 29, 236-245.

Edward L. Paul, Victor Atiemo-Obeng, and Suzanne M. Kresta, ed. (2003), Handbook of Industrial Mixing: Science and Practice, Wiley.

Fasano , J. B. Brodkey, R. S. Haam, S. J.(1991), Local wall heat transfer Coefficients using surface calorimeters, North American Mixing Forum 8.

Jaworski Z., K. N. Dyster and A. W. Nienow (2001), The effect of size, location and pumping direction of pitched blade turbine impellers on flow patterns LDA measurements and CFD predictions, Trans IChemE, Vol. 79, page 25-32.

Jordan, P.J., Neubert F.E (1998), Heat transfer in an unbaffled agitated vessel with a heating jacket over part of the side, International Heat Transfer Conference, Korea.

Jordan, P.J., Neubert, F.C. (1998), Heat transfer in an unbaffled agitated vat with a heating jacket over part of the side, Proceedings of the 11th International Heat Transfer Conference, Kyongju, Korea, Vol. 5,199-204.

Jordan, P.J(1998), Agitation for heat transfer in small tanks, Proceedings of Farm Vat Refrigeration Seminar. Wellington, 37-51.

Karcz J., M. Cudak (2002), Efficiency of the heat transfer process in a jacketed agitated vessel equipped with an eccentrically located impeller, Chemical Papers 56 (6) 382–386.

Khopkar A. R., J. Aubin, C. Xuereb, N. Le Sauze, J. Bertrand, V. V. Ranade(2003), Gas liquid flow generated by a pitched blade turbine particle image velocimetry measurements and CFD simulations, Industrial and Engineering Chemistry Research 42, 5318-5332.

Kumpinsky, E.(1995), Heat transfer coefficients in agitated vessels: Latent heat models, Industrial and Engineering Chemistry Research 35, 938 – 942.

Kurnpinsky, E. (1992), Experimental determination of overall heat transfer coefficient in jacketed vessels, Chemical Engineering Communications, 115, 13–23.

Kumpinsky E. (1995), Heat transfer coefficients in agitated vessels. Sensible heat models, Industrial and Engineering Chemistry Research34, 4571-4576.

Kerim Y. (2003), Numerical investigation of stirred tank hydrodynamics, PhD Thesis.

Li M., White G., Wilkinson D., & Roberts K. J. (2004). LDA measurements and CFD modeling of a stirred vessel with a retreat curve impeller. Industrial and Engineering Chemistry Research, *43*(20), 6534-6547.

Mavros P. (2001)**,** Flow visualization in stirred vessels, Chemical Engineering Science, Vol. 79.

Montante G., G. Micale, F. Magelli, A. Brucato, “Experiments and CFD predictions of solid particle distribution in a vessel agitated with four pitched blade turbines”, Chemical Engineering Research And Design, 79, 1005-1010.

Mununga L., K. Hourigan and M. Thompson(2001), Comparative study of flow in a mixing vessel stirred by a solid disk and a four bladed impeller, Conference Adelaide University, Adelaide, Australia.

Mununga L., K. Hourigan, M. Thompson, S. Johnson(2003), Numerical investigations of discharge flow and circulation flow in an unbaffled mixing vessel agitated by a plain disk, Fluid Mechanics and Thermodynamics , Victoria Falls, Zambia.

Muridhar K, T Sundar Rajan, Narosa Publisher, Computational Fluid Flow and Heat Transfer, New Delhi.

Patankar S.V. (1980), Numerical Heat Transfer and Fluid Flow, Hemisphere.

Raguraman C.M., A. Ragupathy, R. Ramkumar, L. Sivakumar (2010), An effect of blade geometry on heat transfer performance in stirred vessel coal water slurry system using coal gasification, International Journal of Engineering Science and Technology Vol. 2(4), 587-594.

Ranade V.V. and Joshi lB, Flow generated by pitched blade turbines measurements using laser doppler anemometer, Chemical Engineering Communications 81 (1), 197-224.

Ranade, V. V., Computational fluid dynamics for reactor engineering, Academic press.

Ranade, V. V. (1997), An efficient computational model for simulating flow in stirred vessels: a case of rushton turbine, Chemical Engineering Science, 52 (24), 4473 – 4484.

Rahmania L., B. Mebarkib, B. Allaouac, B. Draouid (2013), Laminar flow characterization in a stirred tank with a gate impeller in case of a non Newtonian fluid, Advancements in Renewable Energy and Clean Environment, 36, 418 – 427.

Rahmani L., B. Draoui, B. Mebarki, M. Bouanini, O. Hami (2009), Heat transfer to bingham fluid during laminar flow in agitated tank, Review of Mechanical Engineering, Vol. 3.

Reddy A., Bhavanth Rao, M c Ram Reddy(2012), Experimental estimation of heat transfer coefficients using helical coil in an agitated vessel, International Journal of Engineering Trends and Technology,Volume3.

Shekhar S. Murthy, S. Jayanti (2002), CFD study of power and mixing time for paddle mixing in unbaffled vessels, Trans IChemE, Vol 80.

Stephens G.G., M.R. Mackley (2002), Heat transfer performance for batch oscillatory flow mixing, Experimental Thermal and Fluid Science 25, 583–594.

Tabor G., Gosman A.D., Issa R.I. (1996), Numerical simulation of the flow in a mixing vessel stirred by a Rushton turbine, Institution of Chemical Engineers, 25-35.