5th International Chemical Engineering Congress and Exhibition
Kish Island, 2 - 5 January 2008

Effect of baffle arrangement on turbulent flow field in a stirred tank

Vakili M.H.[1], Nasr Esfahany M.
Department of chemical Engineering, Isfahan University of Technology, Isfahan, Iran

Abstract

Turbulent flow in a baffled mixing tank stirred by an impeller was simulated using CFD. The effect of baffle angles with respect to the vertical direction and position of baffle with respect to the tank wall on flow and turbulent properties has been studied. Multiple reference frame (MRF) technique was used for calculations. In this method flow field is divided into two parts, the inner part co-rotating with the impeller and outer part which is fixed. It was found that baffle edge has considerable effect on some flow parameters such as velocity magnitude, turbulent energy dissipation rate and turbulent kinetic energy.

Keywords: stirred tank, CFD, turbulent flow, baffle, MRF method

Introduction

Mixing in stirred tanks is an important operation which is used for a variety of processes in chemical industries. Operations such as: liquid- liquid contactors, polymerization and crystallization process, mass and heat transfer operations, usually carry out in stirred tanks. A key issue in many industrial processes involving stirred tanks is the quality and homogeneity of mixing. Indeed homogeneity of flow field in a stirred tank has a considerable effect on the product quality. Flow field pattern and turbulent properties of flow such as turbulent kinetic energy and energy dissipation rate, have a direct effect on mixing quality.

Generally most stirred tanks used in the transitional and turbulent flow regimes are equipped with baffles. Usually four flat vertical strip baffles are used in mixing vessels that are installed along the tank wall from bottom to topper head of the tank.

In the unbaffled vessel with the impeller rotating in the center, centrifugal force acting on the fluid moves along circular trajectories with high circumferential velocity creating poor mixing and a vortex is created at the free surface which cause to raise the fluid level at the wall and lowers the level at the shaft. Baffles avoid vortex formation and keep the free surface flat, and provide a condition for a better homogeneity of flow field throughout the tank and therefore improvement in the mixing efficiency is achieved. In a baffle stirred tank also swirling flow is converted into a preferred flow pattern desirable for process objectives, such as axial flow for blending and solids suspension, or radial flow for dispersions[1].

Fletcher D. F. et al investigated the vortex shape in a non-standard partially baffled agitated vessel in the form of a glass-lined, under-baffled stirred vessel using both experimental and numerical approaches for an air/water system [1]. Baffle design and its arrangement in stirred tank have also effect on flow field pattern and turbulent properties of flow. Effect of width and number of baffles in mechanically agitated vessels with standard Rushton turbine impellers has been studied by Lu M. et al [2]. Kimihisa I. et al investigated effect of baffle width on the mixing of liquid and solid particles using a water model for the mechanically stirred vessel [3]. Harris C.

---

[1] P8130292@ce.iut.ac.ir

K. et al reviewed recent progress in the predictions of flow in baffled stirred tank reactors [4]. Burcato A. et al investigated numerical simulations of the flow field in baffled mixing tanks, based on three alternative methods [5].

In the present work we use CFD simulations to study effect of baffle position in the tank (connected to the tank wall or has a distance from wall) and baffle angle with respect of vertical surface passing through two opposite baffles, on the turbulent property and flow field pattern.

Geometry of model

CFD simulations were conducted in a tank, with 30cm height and 20cm diameter that contained liquid water at height of 28cm stirred with a two- blade impeller. This tank equipped with four baffles 28cm in height that cover all of tank, 0.5cm thickness and 1.5cm width. Baffles have 0.5cm distance from tank wall. Impeller has 10cm in diameter, 2cm height and 4cm clearance from bottom of tank. Simulations were conducted for three angles of baffles with respect to radial coordinate,-70,-30,-10, 0, 10, 30, 70 degrees (Figure 1).

Creation of geometry and meshing the model

Gambit v.2 package has been used for creation and meshing the model. Due to use of MRF method for simulation, volume of tank should be divided into two cylindrical zones. Inner zone comprises of impeller and outer zone includes tank walls and baffles. Tetrahedral meshes have been used to mesh the model which creates fine meshes (0.25cm) at zone near the impeller and baffles and larger meshes (0.5cm) elsewhere (Figure 2).

After meshing the model we should define boundary type of system. At first, inner zone and outer zone of tank defined as "fluid1" and "fluid2", respectively. Then all of walls, impeller and baffles defined as a "wall" boundary type with no slip condition and interface area that separates two zones defined as an "interior" boundary type which velocities and velocity gradients in both zones are the same.

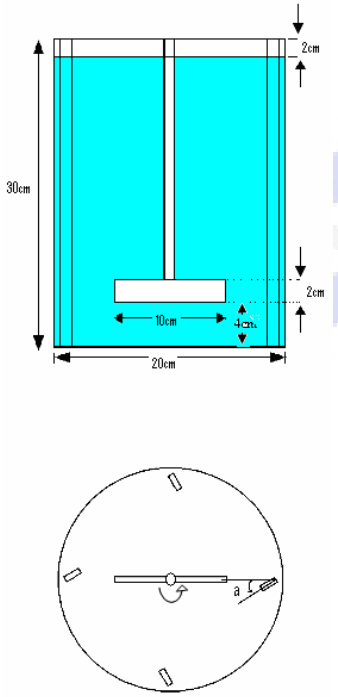

Figure1- geometry of model in axial and radial view

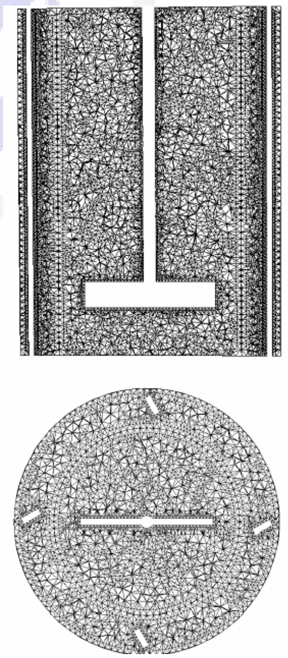

Figure2- tetrahedral mesh of the model in axial and radial view

Governing equations

Governing equations for incompressible fluid flow are as fallowed:

Continuity equation: $$\frac{\partial \bar{u}_j}{\partial x_j} = 0 \qquad (1)$$

Motion equation: $$\frac{\partial}{\partial t}(r\,\bar{u}_i) = -\frac{\partial}{\partial x_j}(r\,\bar{u}_j\bar{u}_i) - \frac{\partial}{\partial x_j}(r\,\overline{u'_j u'_i}) + m\nabla^2\bar{u}_i - \frac{\partial \bar{P}}{\partial x_i} + r\,g_i \qquad (2)$$

In equation (2) $-r\overline{u'_i u'_j}$, Reynolds stress, can be modeled by semi empirical relations. Two-equation k-ε model is used in this work.

Governing equations for turbulent kinetic energy and kinetic energy dissipation rate are [6]:

$$r\frac{\partial k}{\partial t} + r\bar{u}_j\frac{\partial k}{\partial x_j} = t_{ij}^{(t)}\frac{\partial \bar{u}_i}{\partial x_j} - re + \frac{\partial}{\partial x_j}\left[\left(m + m^{(t)}/s_k\right)\frac{\partial k}{\partial x_j}\right] \qquad (3)$$

$$r\frac{\partial e}{\partial t} + r\bar{u}_j\frac{\partial e}{\partial x_j} = C_{E1}\frac{e}{k}t_{ij}^{(t)}\frac{\partial \bar{u}_i}{\partial x_j} - C_{E2}\,r\frac{e^2}{k} + \frac{\partial}{\partial x_j}\left[\left(m + m^{(t)}/s_e\right)\frac{\partial e}{\partial x_j}\right] \qquad (4)$$

Where:

$C_{E1} = 1.44$; $C_{E2} = 1.92$; $C_m = 0.09$; $s_k = 1.0$; $s_e = 1.3$

For liquid in contact with solid surface no-slip condition was used. Zero shear stress was used for free surface. Interior condition was used for interface between fluid1 and fluid2. It means that at the interface the velocities and velocity gradients in both zones are the same.

CFD simulation technique

Multiple reference frame (MRF) method was used for simulations in this work. To use this approach, a rotating coordinate system has been adopted for the inner zone of model named fluid1, which its rotating rate was set equal to impeller agitation rate, and a non-moving coordinate system has been defined for outer zone including baffles named fluid2. Angular velocity of impeller has been set zero with respect to the rotating coordinate system.

To start computations initial guesses should be specified for velocity, pressure and turbulence parameters. The angular velocity of rotating reference frame at the first step of calculations has been set as 5% of actual operating condition. After about 1000 time step, results are saved and used as initial guess for the actual problem. This procedure prevents solution of being diverged. Convergence was achieved when residuals on continuity, velocities, kinetic energy and energy dissipation rate all become less than 10^{-5}.

Results

Flow field in a fully baffled mixing tank stirred by a two-blade impeller as shown in figure1 containing water, was simulated. Impeller rotating speed was 400rpm and Reynolds number was about 400000. Baffles were set at 0.5cm distance from tank wall. Simulations were performed for -70, -30, -10, 0, 10, 30 and 70 degrees of baffle from vertical surface. It was found from simulations that there are three zones that have different turbulent field. A small zone around the impeller that energy dissipation rate of turbulent flow is very large. A zone around the baffles that energy dissipation rate is relatively large, less than impeller zone and finally a relatively, homogenous large zone in fluid bulk, named circulation zone. Figure 3 shows turbulent energy dissipation rate contours at cross sectional surface in 5cm distance from bottom of tank. It is seen that density of contours around the impeller is larger than baffles, and circulation zone is relatively homogenous.

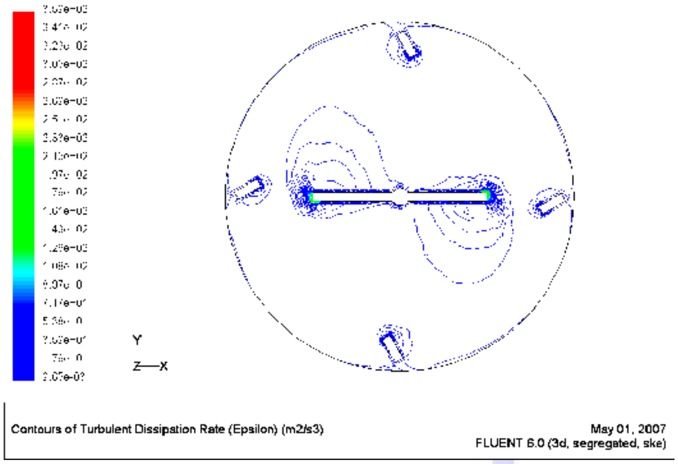

Figure3- contours of energy dissipation rate

Boundaries of these three zones were determined base on energy dissipation gradient. In this method impeller zone begin from impeller tip to point where energy dissipation rate gradient begin to decrease. Baffle zone also begin from baffle edge to point where energy dissipation rate gradient begin to decrease. The remaining fluid is in the circulation zone. Boundary between each two zones determined with an energy dissipation value named energy dissipation rate cut-off, $\varepsilon_{cut,}$ which change in gradient take place on it. Therefore we found two energy dissipation rate cut off in each state that determine between impeller-circulation and baffle-circulation boundaries. Obtained values of energy dissipation rate cut off from energy dissipation rate gradient for different values of baffle angle are shown in table1.

Table1: energy dissipation rate cut off obtained from gradient method

Baffle angle (degree)	$(\varepsilon_{cut})_{imp}$ (m^2/s^3)	$(\varepsilon_{cut})_{baf}$ (m^2/s^3)
-70	61.8	6.51
-30	71.2	10.15
-10	75.6	6.58
0	76.41	6.38
10	82.03	11.2
30	78.9	14.35
70	65	6.19

Effect of baffle angle on velocity

Figure 4 shows velocity vectors at axial surface that pass through middle of impeller and baffles for states that baffles have -30, 0, 30 degrees angle with respect to this surface. It is seen from figures 4-a, 4-b and 4-c that velocity vectors at flow field generally have alike shape in all three states. In all state heading the vectors at the impeller tip is horizontal and into the tank wall and there are two circulating flow at the top and under the impeller, near the baffles. There is some local difference between these states in velocity vector of circulating flow field. At the upper circulating flow, heading the vectors near the baffle is directly to the top of tank for 0 degree of baffle angle whereas it deviates to the center of tank for -30 degree and deviate to the tank wall for the 30 degree. At the lower circulating flow also there is some difference between three states especially at the corner of tank near the baffle. Heading the vectors is horizontal at this zone for -30 and 0 degree of baffle and deviate to the top for 30 degree of baffle angle.

Figure 5 shows velocity vectors at cross section surface which are at 5cm distance from bottom of tank. Concerning to these figure it is seen that by changing the baffle angle there is no considerable change in radial flow field except around the baffles that the direction of flow changes.

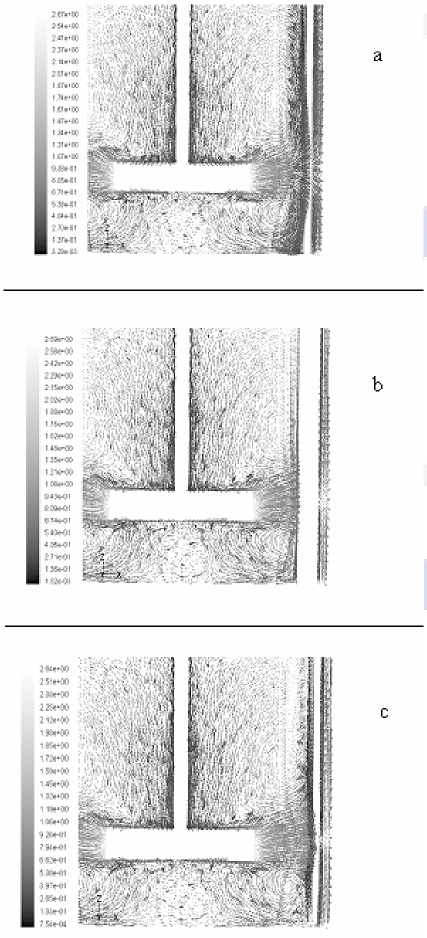

Figure4- velocity vector axial surface that pass through middle of impeller and baffles for state baffle angle a-(-30), b-(0), c-(30) degrees

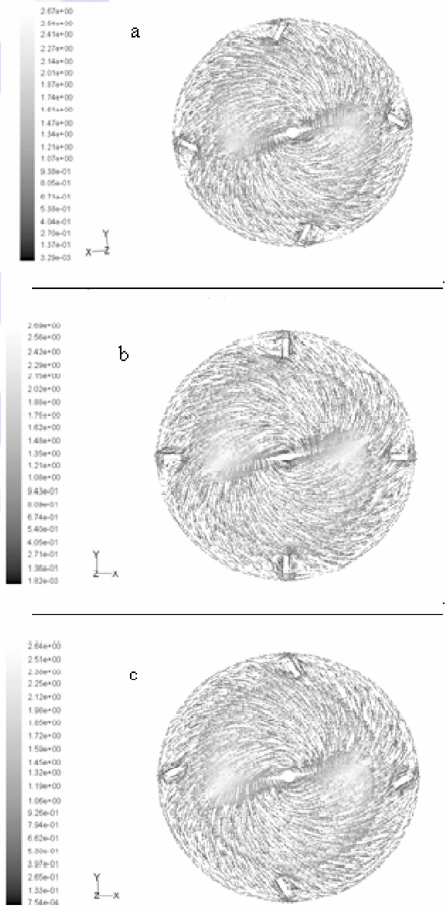

Figure5- velocity vector radial surface that is in 5cm distance from bottom of tank for state baffle angle a-(-30), b-(0), c-(30) degrees

Figure 6 shows variation of velocity magnitude with baffle angle in three zones. It is seen from figure 6-a that at the baffle zone at cases that baffle angle is negative velocity magnitude increase with increasing absolute value of baffle angle. Indeed each baffle deviate from 0 degree to negative value, the area confronts against the fluid flow decrease, therefore velocity magnitude increase. For positive values of baffle angle there is no considerable change with increasing baffle angle at this zone. When baffle angle increase at the positive aspect, although the area confronts against the fluid flow decrease that has an incensement effect on velocity magnitude , but the head of baffles deviate against the flow that has a reducer effect on this parameter. Thus deviation of baffle into the positive aspect has no considerable effect on velocity magnitude. At impeller zone velocity magnitude has no considerable variation with baffle angle. At circulation zone velocity magnitude increases with increasing the absolute value of baffle angle for both positive and negative values. When baffle angle is 0 degree, it is nearest distance of baffles to the fluid bulk and so has maximum effect on fluid velocity at the circulation zone and reduces it. Each baffle angle increase in both negative and positive aspect, distance of baffle to the fluid bulk increase, thus effect of baffle on fluid velocity decrease and velocity increase.

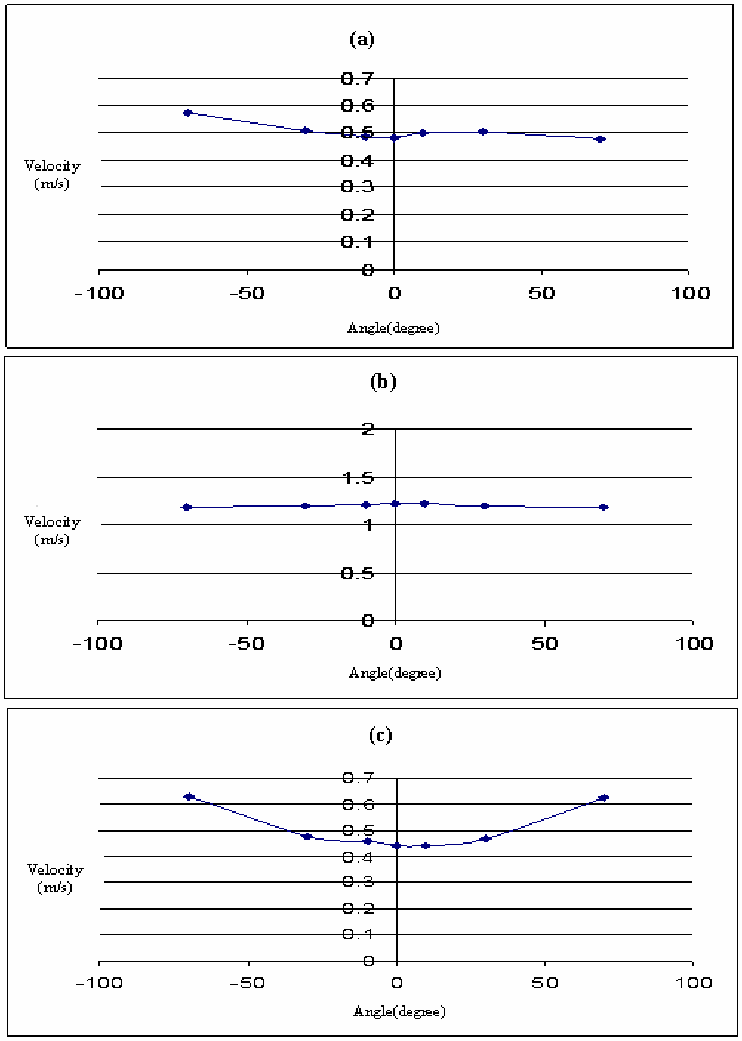

Figure6- velocity magnitude variation with baffle angle at
a-baffle zone, b-impeller zone, c- circulation zone

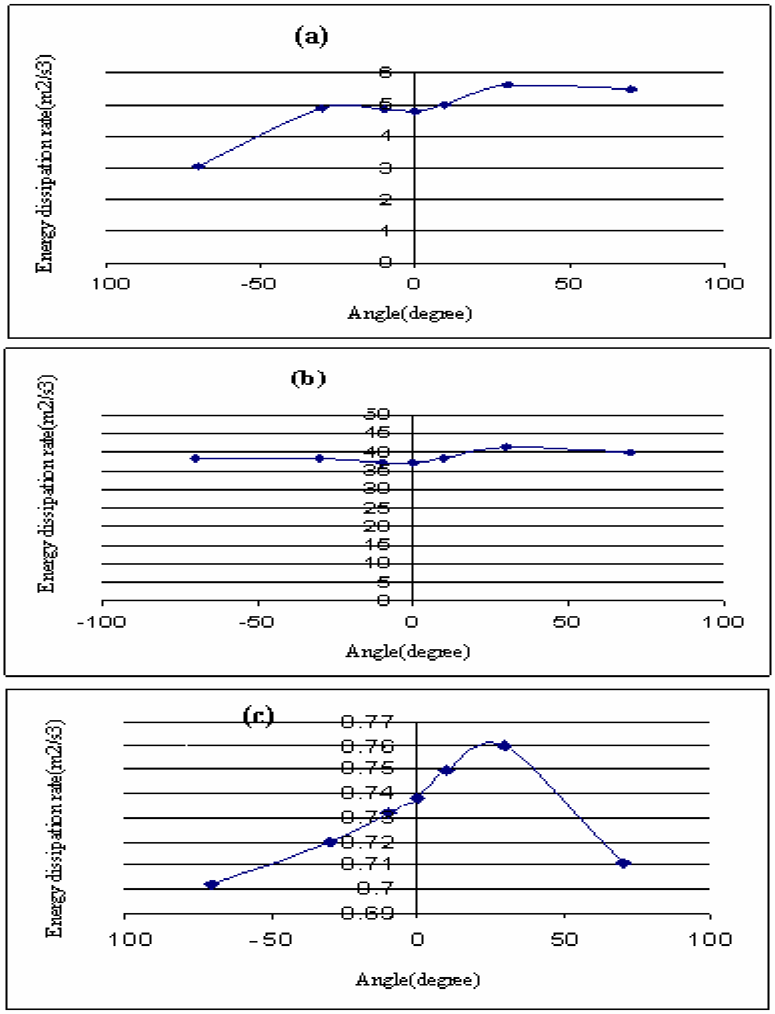

Figure7 energy dissipation rate variation with baffle angle at
a-baffle zone, b-impeller zone, c- circulation zone

Effect of baffle angle on turbulent property

Figure 7 shows variation of energy dissipation rate (mass-weighted average) with baffle angle at three zones. It is seen from figure 7-a that at baffle zone, minimum value of energy dissipation rate increase is in -70 degrees of baffle angle and with nearing to 0 degrees and then up to about 25 degrees energy dissipation rate increase and from 25 degrees no considerable variation was observed. Indeed with variation of baffle angle from -70 to 0 degree the area that flow impact to it increase that lead to increase energy dissipation rate. From 0 to 25 degrees although the impact area of flow to baffle decrease but the edge of baffle deviate against the flow that lead to energy dissipation rate increase (most energy dissipation rate usually carry out at sharp surfaces [8]). It is seen from figure 7-b that baffle angle has no considerable effect on energy dissipation rate at impeller zone.

Figure 7-c shows variation of energy dissipation rate (mass-weighted average) with baffle angle at circulation zone. It is seen that for negative values of baffle angle, energy dissipation rate

decrease with increasing baffle angle for positive values of baffle angle, energy dissipation rate increase with increase baffle angle from 0 to about 25 degrees, and decrease from 25 degrees. Indeed at high value of baffle angle both in positive and negative aspect, baffles are far from bulk flow and there is no strong impact of flow to the baffles, thus energy dissipation rate decrease at circulation zone.

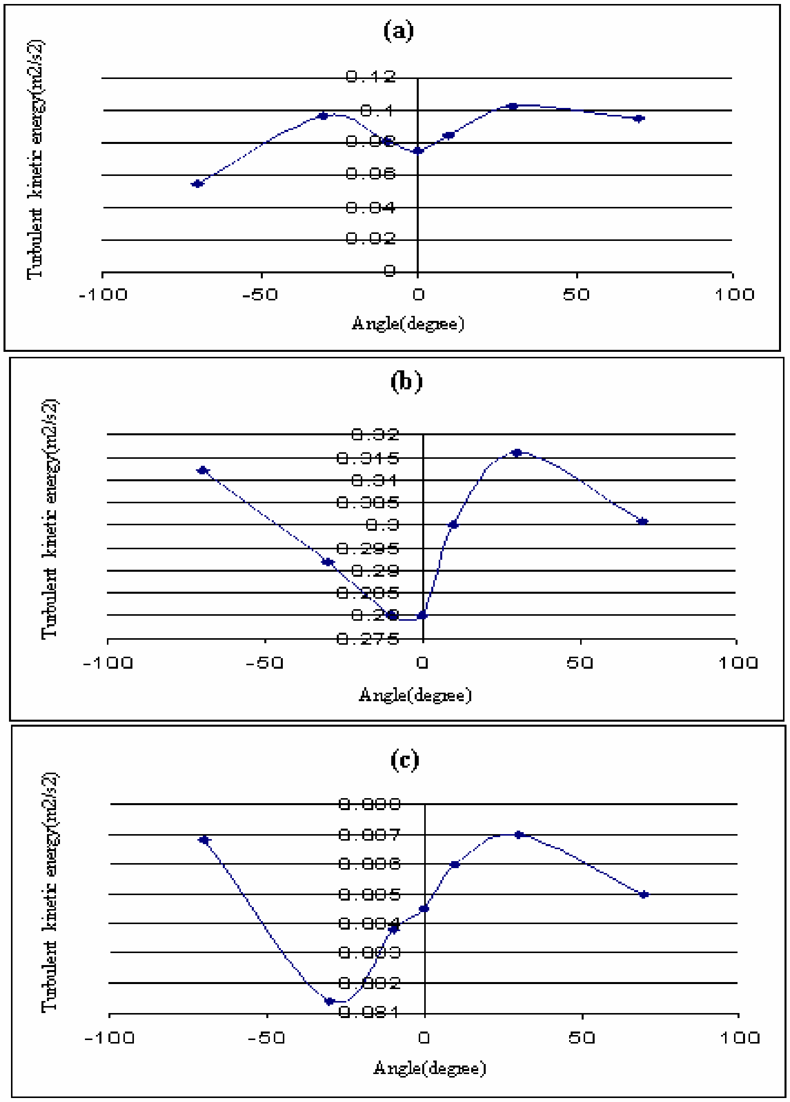

Figure8- turbulent kinetic energy variation with baffle angle at
a-baffle zone, b-impeller zone, c- circulation zone

Figure 8 show variation of turbulent kinetic energy (mass-weighted average) with baffle angle at three zones. It is seen from figure 8-a that there are two maximum values of kinetic energy at -25 and 25 degrees on baffle zone and a relative minimum value at 0 degree. Indeed at 0 degree impact of flow to the baffle lead to generate a reverse flow that reduces momentum transfer and so turbulent kinetic energy decrease. Each baffle angle nearing to the -25 and 25 degree, this reverse flow become weak and lead to turbulent kinetic energy increase. From 25 degrees to upper values baffle nearing to the tank wall that velocity becomes lower and so momentum transfer decrease and so turbulent kinetic energy decrease. It is seen from figure 8-b that turbulent kinetic energy only increase with increasing absolute value of baffle angle at impeller zone. For positive values of baffle angle, there is a maximum value of kinetic energy at 25 degree at this zone.

It is seen from figure 8-c that for negative values of baffle angle, there is a minimum value of kinetic energy at point with baffle angle -25 degrees and for positive values there is a maximum value at point with baffle angle -25 degrees at circulation zone. Impact of fluid flow to the baffle induces a force to the flow that leads to turbulence generation and so increase kinetic energy. At the negative value of baffle angle this force in weaker and so turbulence generation is lower that decrease kinetic energy. But from -25 to -70 degrees it is seen an increase in kinetic energy, it is because of existence of a low energy dissipation in this state that leads to existence of large eddies that can transfer momentum and increase kinetic energy. At positive value of baffle angle the force has induced to the floe is stronger that leads to increase kinetic energy, but from 25degrees to upward baffle go away from fluid bulk and this force become weak and decrease kinetic energy.

Conclusions

It was found from simulations that at the baffle zone at cases that baffle angle is negative velocity magnitude increase with increasing absolute value of baffle angle and for positive values of baffle angle there is no considerable change with increasing baffle angle at this zone. At impeller zone velocity magnitude has no considerable variation with baffle angle. At circulation zone velocity magnitude increases with increasing the absolute value of baffle angle for both positive and negative values.

It was found also that at baffle zone, minimum value of energy dissipation rate increase is in -70 degrees of baffle angle and with nearing to 0 degrees and then up to about 25 degrees energy dissipation rate increase and from 25 degrees no considerable variation was observed. Baffle angle has no considerable effect on energy dissipation rate at impeller zone. At circulation zone for negative values of baffle angle, energy dissipation rate decrease with increasing baffle angle for positive values of baffle angle, energy dissipation rate increase with increasing baffle angle from 0 to about 25 degrees.

Notation

C_{E1}: Experimental coefficient in equation ε
$C_{E2:}$ Experimental coefficient in equation ε
g: Gravity (m/s^2)
k: Turbulent kinetic energy (m^2/s^2)
$\overline{P}$: Average pressure (Pa)
t: Time variable (sec)

$\overline{u}$: Average velocity in x direction (m/s)
u' : Fluctuation velocity (m/s)

Greek
ε: Turbulent energy dissipation rate (m^2/s^3)
μ: Viscosity (kg/m.s)
$\mu^{(t)}$: Turbulent viscosity (kg/m.s))
ρ: Density (kg/m^3)
σ_k :Experimental constant in equation k
σ_ε : Experimental constant in equation ε
$\tau_{ij}^{(t)}$: Turbulent momentum flux (Reynolds stress)

References

1. Torrea J.P., Fletcherb D.F., Lasuyec T., Xuereba C., (2007)" An experimental and computational study of the vortex shape in a partially baffled agitated vessel", Chemical Engineering Science 62, 1915 – 1926.
2. Lu W.-M., Wu H.-Z., Ju M.-Y., (1997)" Effects of baffle design on the liquid mixing in an aerated stirred tank with standard Rushton turbine impellers", Chemical Engineering Science, 52(21), 3843-3851.
3. Kimihisa I., Norihiro Y., Seiichiro K., (2006)" The Evaluation of the Mixing Properties of the Mechanically Stirred Cylindrical Vessel", ISIJ International, 46(12), 1791-1794
4. Harris C.K., Roekaerts D., Rosendal F.J.J., Buitendijk F.G.J., Daskopoulos Ph., Vreenegoor A.J.N., Wang H. (1996), "computational fluid dynamics for chemical reactor engineering", Chemical Engineering Science, 51(10), 1569-1594.
5. Burcato A., Ciofalo M., Grisafi F., Micale G., (1998) "Numerical prediction of flow fields in baffled stirred vessels: A comparison of alternative modeling approaches", Chemical engineering science 53(21), 3653-3684.
6. Patel, V.C., Rodi, W., Scheaeres, G. (1985)," Turbulence models for near-wall and low Reynolds number flows", AIAA journal, 23, 1308-1319.
7. Alexopoulos, A.H., Maggioris, D., Kiparissides, C., (2002) "CFD analysis of turbulence non-homogeneity in mixing vessels. A two-compartment model" Chemical Eng. Science 57, 1735-1752.
8. Nasr Esfahany M., Vakili M.H. ," CFD modeling of turbulent flow in a baffled stirred tank, comparison of turbulent flow characteristics at three different zones" submitted.