Proceedings of the World Congress on Engineering and Computer Science 2007
WCECS 2007, October 24-26, 2007, San Francisco, USA

Computational Fluid Dynamic Analysis of Mixing Characteristics inside a Jet Mixer for Newtonian and Non Newtonian Fluids

K. Sendilkumar, P. Kalaichelvi*, M. Perumalsamy, A. Arunagiri and T. Raja

Abstract- **Jet mixers with side entering jets and their different configurations are used for mixing purposes, e.g., homogenization of physical properties and composition, prevention of stratification or deposition of suspended particles, for improved rates of heat, mass transfer and chemical reaction to achieve faster mixing time. Most of the researchers have focused on experimental estimation of mixing time and proposing suitable correlations for the prediction of mixing time and recent ones being on flow visualization. Though there are few researcher works on flow visualization, many of them have not validated their results with the real time values. Hence it is proposed to study the flow pattern inside the jet mixer under different configuration. A 3D jet mixer (cylindrical tank with jet of different configuration and aspect ratio of 1) was modeled using commercially available software Fluent. This model was simulated for different sets of nozzle configurations and the simulated mixing characteristics were obtained for both Newtonian and Non-Newtonian (9% carboxyl methyl cellulose (CMC) solution) fluids. A detailed comparative analysis has also been made with the already estimated experimental mixing times for Newtonian fluids. The simulated mixing characteristics of 9% CMC solution (Non-Newtonian fluid) have been compared with water (Newtonian fluid) mixing characteristics under the same operating conditions.**

Index Terms--Computational Fluid Dynamics, Jet Mixing, Modeling and Simulation

The manuscript revised on 6th August 2007

P. Kalaichelvi, M. Perumalsamy and A. Arunagiri are with Department of Chemical Engineering, National Institute of Technology, Tiruchirappalli – 620 015, Tamil Nadu, India (kalai@nitt.edu, mpsamy@nitt.edu and aagiri@nitt.edu)

K. Sendikumar was with Department of Chemical Engineering, National Institute of Technology, Tiruchirappalli – 620 015, Tamil Nadu, India.

T. Raja is with Honeywell Technology Solutions (HTS), Bangalore-560076,India. (raja.thiruvengadam@honeywell.com)

I. INTRODUCTION

In the Jet age of today, jets should find their uses in several places and mixing which is basically outcome of fluid flow is no exception. Jet mixers are simple and require close to zero maintenance. This jet mixing systems rely on the pumping energy and therefore viscosity of the liquid with respect to the available pressure is the only limitation. Jets have almost thrown side entry agitator out of use. The jet is designed to create appropriate velocities. It is then placed in the tank at a right position and with a correct angle so that appropriate swirl and vortex is created for the bulk mixing. In some situations more than one jet are required. Jets with Ejectors and with or without static mixers have been effectively used in several gas liquid applications.

Mixing is an important unit operation in many chemical engineering applications. It can be used for a variety of purposes, e.g., homogenization of physical properties and composition, prevention of stratification or deposition of suspended particles, for improved rates of heat, mass transfer and chemical reaction. Mixing using jets of liquids has the advantage of having no moving parts inside the reactor. Here, a part of the contents of the vessel is circulated by drawing it through a pump and returning it at high speed through a nozzle. The resulting jet of liquid entrains some of the surrounding liquid and creates a circulation pattern within the vessel thus leading to mixing of the contents. Jet mixing nozzles are a single phase submerged mixing devices.

Various techniques have been used by researchers to examine the performance of impinging jets in an attempt to achieve a fundamental understanding of mixing in these systems. These include optical techniques and measurements of conductivity or temperature. There have been many correlation proposed to calculate the mixing time for a jet mixer. These correlations can be classified into two main categories: one show the dependence of mixing time on the jet Reynolds number, Re_j *while* the other category does not. The jet Reynolds number Re_j *is* calculated as $Re_j = \rho\, d_j V_j / \mu$, where d_j is diameter of the jet and V_j is the velocity of jet at the jet inlet. Fosset and Prosser [1] employed a conductivity technique to measure mixing time in a jet mixer. Their original correlation included only terms for tank diameter, jet diameter and jet velocity. It shows no dependence on Re_j.

$$t_{95} = 9\left(\frac{D^2}{d_j V_j}\right) \qquad (1)$$

Where t_{95} is the mixing time, D is the tank diameter, d_j is the jet diameter and V_j is the jet velocity. The above equation is applicable to the turbulent jet regime only.

Coldrey [2] proposed that, for side entry jet mixing, a longer jet length produces a more effective mixing jet and therefore reduces mixing time. Coldrey formulated a mixing time equation independent of the jet Reynolds number for turbulent jet regimes.

An improved correlation offering a better fit of blend time data for turbulent jet mixed vessels was proposed by Grenville and Tilton [3]. They assumed that the mixing rates at the end of the jets free path, estimated from the turbulent kinetic energy dissipation rate, and controls the mixing rate for the whole vessel. This is shown as

$$t_{99} = \left(\frac{X}{d_j}\right)^2 \left(\frac{d_j}{V_j}\right) \qquad (2)$$

Where as t_{99} is 99% mixing time, X is the jet path length. They predicted that, for a given volume, an optimum geometry exists for a jet mixed vessel, which allows desired blend time to be achieved for minimum power input. Grenville and Tilton correlation showed a better fit of the experimental data.

Fox and Gex [4] indicated that mixing time is dependent on the jet Reynolds number; the dependence being strong in the laminar jet regime but slight in the turbulent jet regime. Cziesla et al. [5] simulated jets impinging on a wall using large eddy simulation (LES) simulation technique. Souvaliotis et al. [6] investigated errors and limitations of mixing simulations. They identified and examined errors due to discretization, time integration, and round-off. They concluded that accurate quantitative information can only be obtained from numerical simulations if certain proper steps such as mesh refinement are taken into consideration. In summary, it can be said that, until recently, little was known about the fluid flow, velocity field and detailed mixing characterization of jet mixers. As a result, impinging jet mixers have thus far been designed individually for each specific process, and the optimization of such systems has been done largely by trial and error. Some studies have recently investigated mixing in jet agitated mixing tanks; however, no study has so far investigated thoroughly the effects of the angle of the jet and the elevation of the jet on mixing time. The present study investigates the effects of jet angle and elevation on mixing in a fluid jet agitated tank using CFD. The results shed significant light on the velocity field and mixing characterization involved in jet mixers. This will be explained in the following sections.

Lane and Rice [7] investigated liquid jet mixing employing an inclined side entry jet. They studied tow designs for inclined side entry jet mixing. They formulated a general expression to correlate mixing time in terms of jet velocity, tank dimensions and fluid properties. They proposed a mixing time formula that can be used to predict the time required to achieve a 95% degree of mixing throughout an inclined side entry jet mixer. They introduced a mixing time factor and showed that it is function of jet Reynolds number.

Jayanthi [8] used a general-purpose computational fluid dynamics (CFD) code, CFX, to investigate the hydrodynamics of jet mixing in cylindrical vessels. He studied the effects of the flow circulation patterns within the reactor and their effects on the mixing of a soluble salt. He showed that a key factor in reducing mixing time is minimizing or eliminating dead zones in the reactor and he recommended the use of conical bottom for the tank as a possible improvement. Patwardhan [9] presented a CFD model of jet mixed tanks which effectively predicted overall mixing times. However poor agreement was observed between the numerical and experimental results of concentration profiles as function of time at various locations. Zughbi and Rakib [10]-[12] presented a CFD model of mixing in fluid jet agitated tank and validated their numerical model against the experimental results of Lane and Rice. Raja et al. [13] have optimized the shape of the jet mixing tank base using CFD and also validated with their experimental mixing values.

In summary, it can say that, until recently little was known about the fluid flow, velocity field and detailed mixing characterization of jet mixers. As a result, impinging jet mixers have thus far been designed individually for each specific process, and the optimization of such systems has been done largely by trial and error. Some studies have been investigated mixing in jet agitated mixing tanks; however, so far no study has thoroughly investigated the effects of the jet angle and elevation on mixing time. The present study investigates the effect of jet angle, elevation of jet and effect of nozzle diameter on mixing in a fluid jet agitated tank using CFD.

II. EXPERIMENTAL SET UP AND ITS CFD MODELING

A. *Experimental set up*

The jet-mixing tank used in the reported work is a cylindrical tank of 0.28 m internal diameter with a working fluid height of 0.28 m and total height of the tank 0.45 m. There is a gap above the liquid surface, which is open to atmosphere. A pipe diameter of 0.025 m was used to draw the liquid from the bottom of the tank through a centrifugal pump and returned to the contents of the tank through side entry jet as shown in the Fig. 1.

Tap water was used as a working fluid and sodium chloride is an electrolyte to find the conductivity and hence to find the mixing time. The time taken for reaching 95 % of the final conductivity value was taken for the complete mixing time.

The experiments were carried out to find the mixing time by varying the nozzle angle from 15^0 to 60^0, nozzle diameter from 0.005 m to 0.015 m and nozzle positions as top, $2/3^{rd}$ from the bottom and bottom.

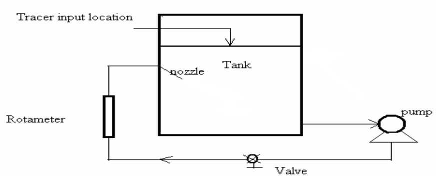


Fig.1. Experimental set up

B. *CFD modeling*

The governing equations for a general mixing problem are the mass, momentum and energy equations. The numerical solutions of transport equations mainly comprise discretization of the governing equations. In this study, a general purpose CFD package, FLUENT was used to solve the equations using finite volume approximations. For the present study residual value for energy was set at 10^{-6} and the residual value for all other variables was set at 10^{-3}. The RNG k-epsilon model was used to characterize turbulent motion and the higher-order upwind differencing scheme was for discretization.

C. *Grids*

A tetrahedral mesh was used to discretize the computational domain. A mesh interval of 1mm was used for the tank. A total of 130800-130900 cells were used to mesh the tank. This mesh is fine compared with the meshes used by other researchers [9]-[11]. For the present study, the three-dimensional flow domain which consists of an x-y-z plane, y being the axial direction and x and z being the radial direction.

D. *Calculation strategy*

The progress of the calculations was as follows. The steady state velocity field was obtained first for the given geometry, inlet and outlet conditions. This was then given as initial velocity field for the transient calculations. The initial concentration was to zero throughout the flow domain. The tracer is injected for a time period of 4 s. The evolution of the concentration field with the introduction of the tracer was then calculated by marching forward in time with a time step of 0.1s which was increased to a maximum of 1 s. To represent the effect of recirculation and consequent time-varying inlet concentration, the calculations were carried out in short periods during which the inlet and outlet concentrations were fixed. At the end of each period, the mean outlet concentration was added to the inlet concentration. To represent the effect of feeding back without any time delay, the fluid going out through the outlet. For calculating the mixing time in the simulated model, probes were kept at 5 different positions to monitor the concentration in that position. The probe locations are presented in Table I, when the concentration in the probes remains unchanged with time, the corresponding time is taken as the mixing time for the system. The calculation was

TABLE I
PROBE POSITION IN THE SIMULATED MODEL

Probe	X axis (m)	Y axis (m)	Z axis (m)
1	0.00	0.25	0.00
2	-0.09	0.20	-0.09
3	-0.09	0.15	0.09
4	0.09	0.10	-0.09
5	0.09	0.04	0.09

carried out repeatedly in this manner until the concentration at all points differed by less that 0.01 % from the fully mixed value.

III. RESULTS AND DISCUSSIONS

A. *Newtonian fluid (Water)*

A three dimensional experimental set up consisting of a cylindrical vessel of diameter 0.28 m, with a flat bottom having liquid level to a height of 0.28 m (aspect ratio = 1) has been modeled and simulated in fluent (6.1.22) to obtain the velocity and concentration profiles for water as the test fluid. The concentration was measured by positioning probe at different points in the tank as described above, when the concentration in each probe remains unchanged with time and that time is taken as the mixing time. The flow patterns are used to describe the process of mixing in jet mixer under different operating conditions like nozzle location, nozzle angle and nozzle diameter. The results obtained are analyzed, compared with the experimental results and discussed under the following headings.

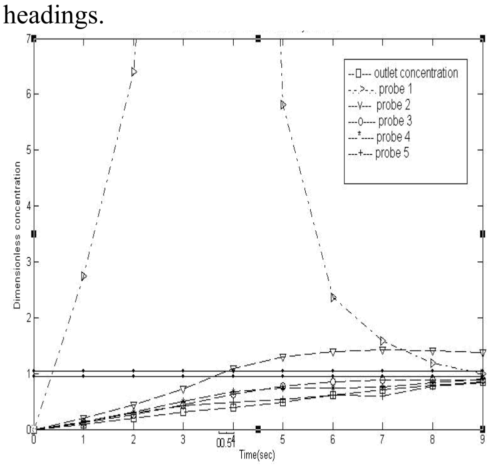
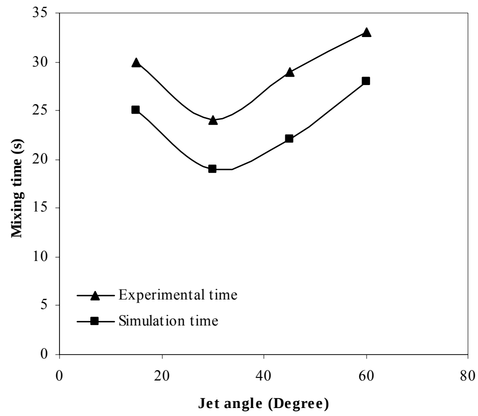
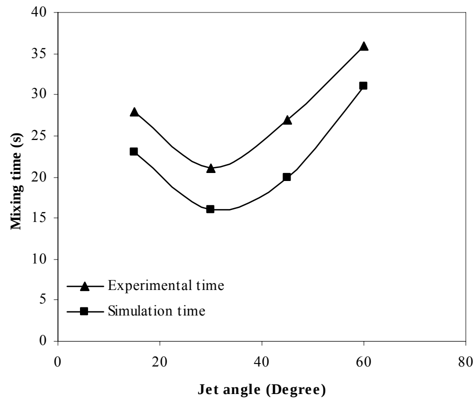


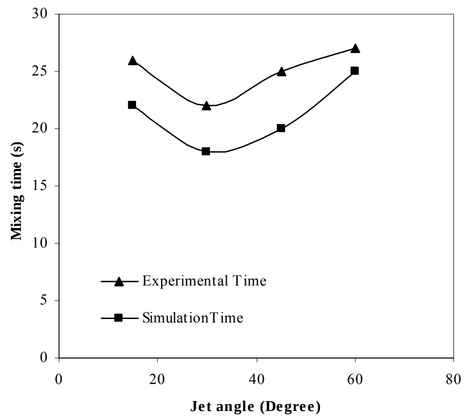
Fig.2. Simulated dimensionless concentration profile for the nozzle placed at the top position with an angle of 15 Degree for a velocity of 8m/s (water).



(a)



(b)



(c)

Fig. 3. Comparison of experimental time vs. simulation time for nozzle diameter of 0.005 m for a velocity of 8 m/s (water) placed at (a) top position (b) $2/3^{rd}$ position (c) bottom position

B. *Effect of nozzle angle*

The effect of nozzle angle on mixing time is studied by varying the nozzle angle and maintaining nozzle diameter, nozzle position and velocity of jet as constants. The dimensionless concentration (Concentration at point/ fully mixed concentration) for a nozzle diameter of 0.005 m positioned at the top of the tank with a velocity of 8 m/s is shown in Fig. 2. The comparison of simulation and experimental mixing time for a set nozzle configuration is shown in Fig.3. (a), Fig.3. (b), and Fig.3. (c). The simulated velocity vector for nozzle placed at bottom position with an angle of 45^{o} and for a velocity of 8 m/s is shown in Fig. 4. We observe from both the experimental and simulation results, 30^{0} of nozzle angle is reported to be the best at all the nozzle locations. Fig. 5 shows that 30^{0} gives the shortest mixing time compared to other nozzle angles, the longest mixing time is when the nozzle angle is 60^{0}. This is because the flow patterns s not effective to agitate the contents in the tank as the jet rollover takes place, after wall impingement. For the fixed height of a liquid in a tank, varying the angle of the jet will affect not only the effective mixing length of the jet, but also the overall flow patterns inside the tank, which is the key factor in deciding the overall mixing time.

When the jet angle is further increased from 45^{0} to 60^{0} the mixing time goes on increasing, this due to the fact, a short circuiting phenomenon is observed for the jet in addition to the strong wall effects that again come into effect when the jet is in close proximity to a side wall.

C. Effect of nozzle position

To analyze the effect of nozzle location on flow pattern, simulations are carried out with the developed model for all the three nozzle locations by keeping the nozzle diameter; nozzle

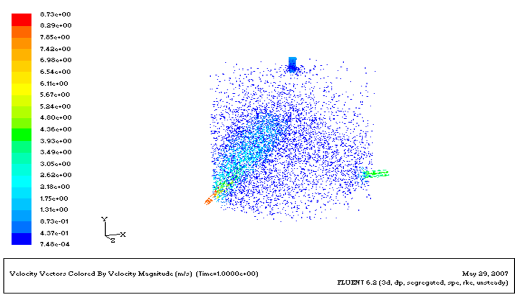


Fig.4. Simulated velocity vectors for nozzle placed at bottom position with an angle of 45 Degree for a velocity of 8 m/s (water).

angle and fluid velocity constant for comparison purposes. The simulation results are compared with the experimental results and shown in .

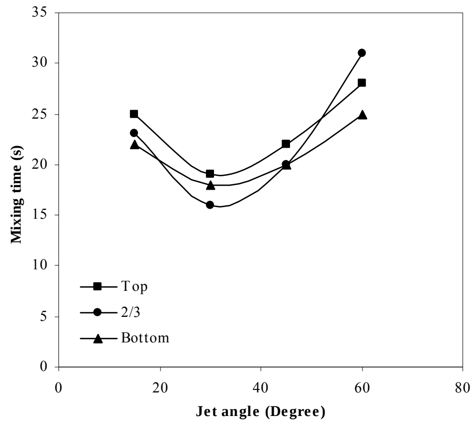


Fig.5. Simulation time for nozzle placed at different position with the nozzle diameter of 0.005 m for a velocity of 8 m/s (water).

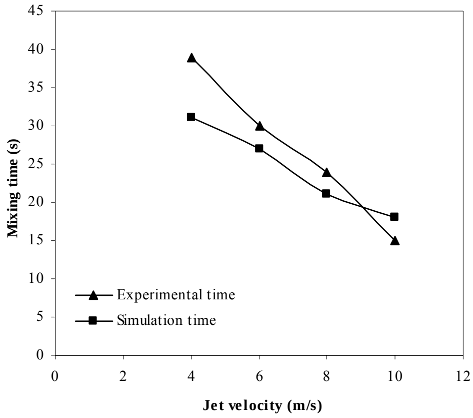
Fig.6. (a), Fig.6. (b) and Fig.6. (c). From the simulation results, the following observations are made

For the nozzle located at the top, shown in Fig.6. (a) when the velocity of the jet is increased, and if the angle formed by the jet with the tank is very less, the jet flows on the liquid surface and impinges on the wall of the tank, while impinging, it losses some of the momentum and flows down the wall of the tank and remaining momentum associated with the jet is used to agitate the fluid in the tank, as clearly shown in Fig.7. For the Nozzle positioned at the top, nozzle angle of 30^{0} gives better mixing as explained in earlier section.

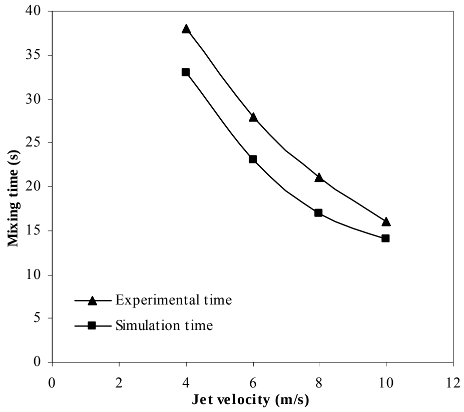
In this case the nozzle placed in the bulk of the liquid (i.e.) two by third from the bottom of the tank, as shown in Fig.6. (b) increasing the velocity increases the turbulence in the tank. This is because all the momentum with the jet is used for agitating the fluid in the tank, and also the head given by the liquid is less compared to the jet placed in bottom position.

When the nozzle is placed at the bottom position of the tank as shown in Fig.6. (c), increasing the velocity increases the turbulence in the tank because it has to overcome the head given by the liquid in the tank. Therefore only a fraction of the momentum with the jet is used to overcome the liquid head and the remaining fraction of the momentum is used to agitate the fluid in the tank.

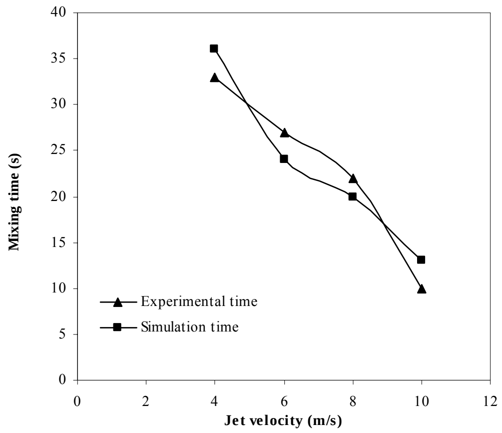
When the nozzle is placed at the bottom position of the tank as shown in Fig.6. (c), increasing the velocity increases the turbulence in fraction of the momentum with the jet is used to



(a)



(b)



(c)

Fig.6. Comparison of experimental time vs simulation time for nozzle diameter of 0.005 m for an angle of 30 Degree (water) placed at (a). top position (b) 2/3rd position (c) bottom position

overcome the liquid head and the remaining fraction of the momentum is used to agitate the fluid in the tank.

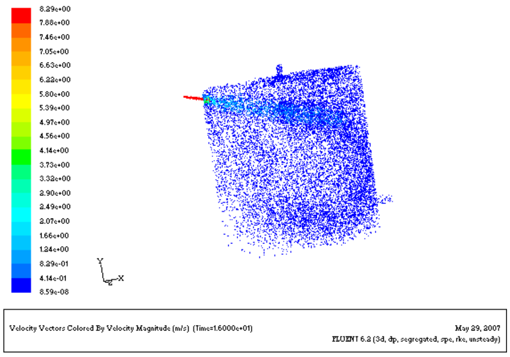


Fig.7. Simulated velocity vectors for nozzle diameter of 0.005 m placed at top position with an angle of 30 Degree for a velocity of 8 m/s (water).

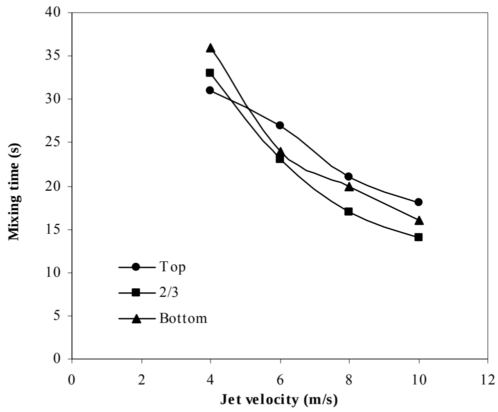


Fig. 8. Comparison of simulation time for nozzles placed at different location with the nozzle diameter of 0.005 m for an angle of 30 Degree (water).

The jets were placed at three different positions (i.e.) at the top of the liquid surface, two-third (from the bottom of the tank) and at the bottom position and simulation results are compared and shown in Fig 8. The jet placed in 2/3rd position gives less mixing time compared to other positions. The jet length is no doubt an important parameter controlling the blending time, but at the same time, the rollover effect of the jet after hitting the tank walls and therefore the flow patterns, are also major parameters in determining the mixing time. This figure also shows that increasing the velocity results in the reduction of mixing time.

D. Effect of nozzle diameter

The developed three dimensional jet mixer model was simulated for all the three nozzle

ISBN:978-988-98671-6-4

diameter by keeping nozzle angle and fluid velocity constant. For the above mentioned conditions, simulations results are carried out by positioning the nozzle location at top, 2/3rd and bottom of the tank. Comparison of experimental time and simulation time for nozzle placed at top position and for angle of 30^o with a velocity of 8m/s and are shown in Fig. 9.

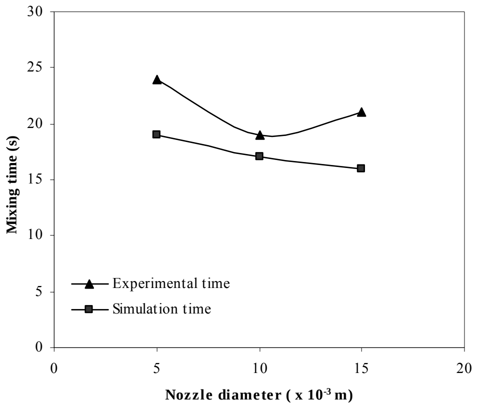


Fig. 9. Comparison of experimental time vs. simulation time for nozzle placed at top position and for angle of 30 Degree with a velocity of 8 m/s (water).

As the nozzle diameter increased it leads to reduction in the mixing time, hence good flow patterns inside the tank. When the diameter is increased the flow rate through the nozzle increases for the same level of velocity. This means that, the liquid is circulated faster through the bigger nozzle, which leads to reduction in the mixing time.

The effect of nozzle diameter on mixing time is obtained by performing the simulation at different nozzle angle at constant velocity and this is shown in Fig. 10.

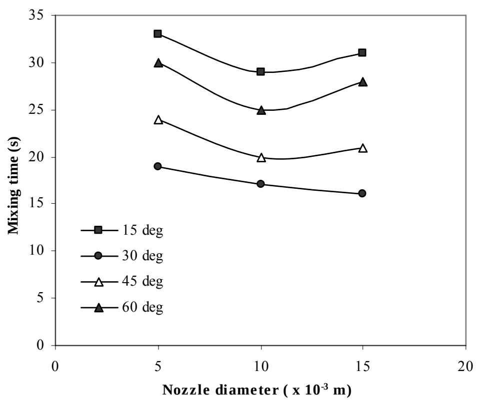


Fig 10. Simulation time for nozzle placed at bottom position and at different nozzle angle with a velocity of 8 m/s (water)

E. *Non Newtonian fluid (9% CMC solution)*

The experimental set-up which was modeled for water was also used to study the flow pattern for 9 % CMC solution. Simulations were carried out for 9 % CMC solution for different nozzle angles (15^0, 30^0, 45^0 and 60^0) by having the nozzle at different position (i.e. top, 2/3 and bottom) and at different velocities (4, 6, 8 and 10 m/s). Concentration profiles resulted during simulations are used to estimate the mixing time at the above said nozzle configuration.

F. *Effect of nozzle angle*

Simulations are carried out for 9 % CMC solution; power law of fluid to define its Non Newtonian properties. The simulated dimensionless concentration profiles [15] for a nozzle diameter of 5mm positioned at the top of the tank, 2/3rd and bottom of the tank with a velocity of 8m/s are used to get the mixing time. Fig. 11 shows the simulated mixing time versus nozzle angle for three different nozzle positions and for a nozzle diameter of 5mm. Shortest mixing time is obtained when the nozzle angle is 30^0 when the nozzle is placed both at 2/3rd and bottom position, same as that of water. This is because the velocity profile is same as that of water but differs in their magnitude.

G. *Effect of nozzle position*

From the Fig.12, we can clearly conclude that the nozzle placed at the 2/3rd position gives the shortest mixing time, this position gives a better circulation pattern inside the tank. The reason for this is same as that of water.

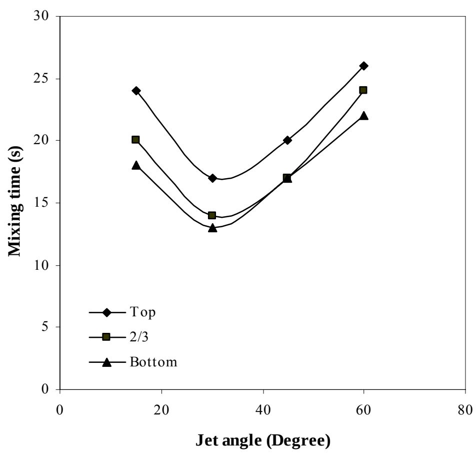


Fig.11 Simulation time for nozzle placed at different position with a diameter of 0.005mm for a velocity of 8 m/s (9% CMC solution).

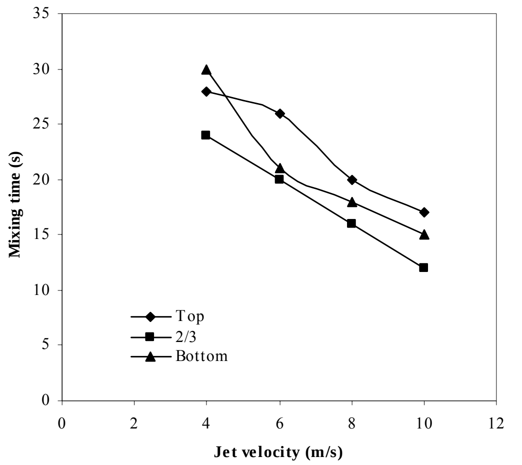


Fig.12. Simulation time for nozzle placed at different location with the nozzle diameter of 0.005 m for an angle of 30 Degree (9% CMC solution).

H. Effect of Nozzle diameter

Fig.13 shows that increasing the nozzle diameter decreases the mixing time, which is due to the increase in mass flux through the nozzle of larger diameter. Nozzle angle of 30^0 gives better mixing time compared to that of other Nozzle angles.

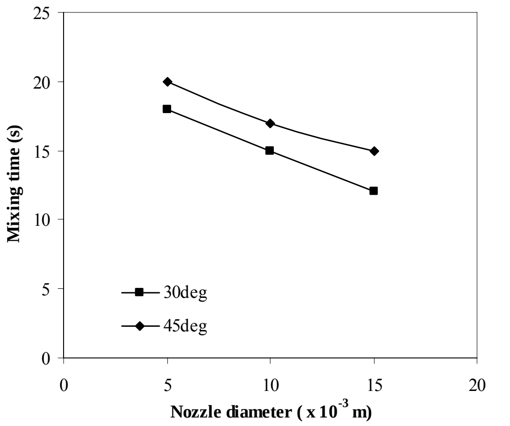


Fig.13. Simulation time for nozzle placed at bottom position with the nozzle diameter of 0.005 m for velocity of 8 m/s (9% CMC solution).

I. Comparison of mixing pattern for water (Newtonian) and 9% CMC solution (Non Newtonian)

Simulated mixing time values for water are compared with 9% CMC solution in Fig. 14. We observe mixing times for 9% CMC solution are lower than the mixing times of water.

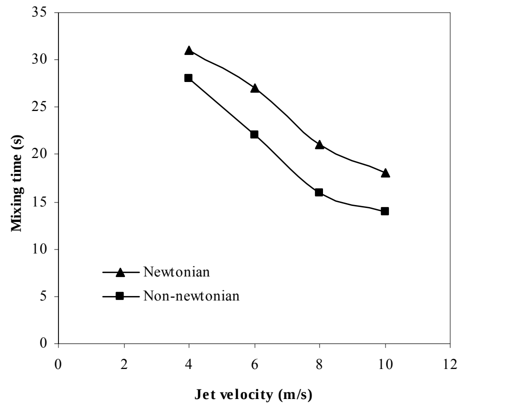


Fig. 14. Comparison of simulated mixing time for water (Newtonian) and 9% CMC solution for the nozzle position at bottom with angle of 30 Degree (Non Newtonian fluid).

Tracer diffuses faster in 9% CMC solution when compared to that of water (Fig. 15) and Fig. 16, which also coincides with the results shown in Fig. 14.

Carboxyl methyl cellulose solution is a shear rate thinning fluid whose viscosity reduces with the intensity of mixing, so the mixing time is less compared to that for Newtonian fluid. The velocity pattern for water and 9% CMC solution (Newtonian and non-Newtonian) are same but the magnitude of velocity is more for water compared to 9% CMC solution. Mixing time for water is 15-24% more than for 9% CMC solution.

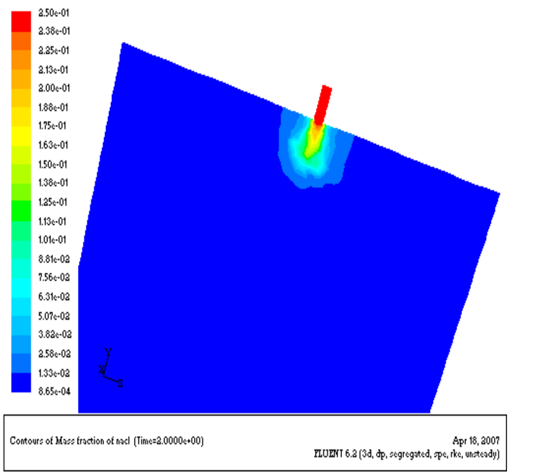


Fig.15. Contours of mass fraction of NaCl for 9% CMC solution after 1 s for the nozzle placed at the bottom position with an angle of 15 Degree for a velocity of 8m/s

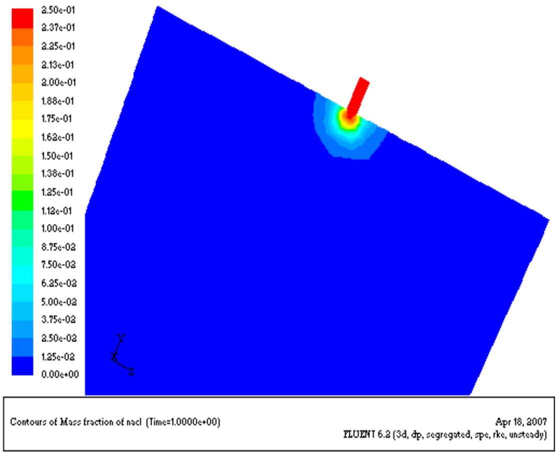


Fig.16**.** Contours of mass fraction of NaCl for water after 1 s for the nozzle placed at the bottom position with an angle of 15 Degree for a velocity of 8 m/s.

IV CONCLUSIONS

The experimental set up used in the previous study (14) was modeled in three dimensions and simulated by varying the configuration viz, the nozzle position, nozzle angle nozzle diameter and velocity of the jet. The effect of individual parameter on flow pattern was analyzed and also compared with the reported experimental results. The same model was used to carry out simulation studies with 9% CMC solution and the simulated results in terms of velocity profile and concentration profile were analyzed and also compared with the simulated results of water.

(1) For water, the nozzle positioned at 2/3rd of the tank could give better flow pattern for angles of inclination of 30^0 and 45^0. The shortest mixing time is observed, when the nozzle angle 30^0 which coincides with the results of H.D.Zughbi and M.A.Rakib [11]. Based on reported experimental and simulated mixing time, an angle of 30^0 was optimized for all the three Nozzle locations.

(2) Dimensionless concentration analysis along with flow pattern analysis could reveal that the nozzle positioned at 2/3rd is the best configuration for Newtonian fluid, which also agrees with reported experimental mixing time reported by Swarnalatha [14]. However the simulated mixing time values are lower than the experimental mixing times under all the nozzle configurations, which may be due to the location of probes considered for simulation study.

(3) Simulation results for the 9% CMC solution show that a nozzle angle of 30 ^{o} at 2/3 rd position is the optimum for the present configuration.

(4) The mixing time for 9% CMC solution is 15-24% less than for water under all nozzle configurations.

Similar simulation analysis has to be performed with other types of Non-Newtonian fluids.

REFERENCES

1. H. Fossett and L. E. Prosser, "The application of free jets to the mixing of fluids in bulk", *Journal of Institute of Mech. Eng.*, vol. 160, 1949, pp. 224–232.
2. P. W. Coldrey, "Jet mixing", unpublished.
3. R. K. Grenville and J. N. Tilton, "A new theory improves the correlation of blend time data from turbulent jet mixed vessels", *Trans. Inst. Chem. Eng.*, vol. 74A, 1996, pp. 390–396.
4. E. A. Fox, and V. E. Gex, "Single phase blending of liquids", *A.I.Ch.E. J.*, vol. 2, 1956, pp. 539–544.
5. T. Cziesla, G. Biswas, H. Chattopadhyay and N. K. Mitra,, "Large-eddy simulation of flow and heat transfer in an impinging slot jet", *Int. J .Heat and Fluid Flow*, vol. 22, 2001, pp. 500–508.
6. A. Souvaliotis, S. C. Jana,, and J. M. Ottino, "Potentialities and limitations of mixing simulations", *A.I.Ch.E. J.*, vol. 41, 1995, pp. 1605–1621.
7. A. G. C. Lane, and P. Rice, "An experimental investigation of liquid jet mixing employing a vertical submerged jet", *Ind. Chem. Engg. Symposium Series,* vol. 64, 1981, pp. K1–K14.
8. A. G. C. Lane and P. Rice, "An investigation of liquid jet mixing employing an inclined side-entry jet", *Trans. Inst. Chem. Eng.s*, vol. 60, 1982, pp. 171–176.
9. S. Jayanti,, "Hydrodynamics of jet mixing in vessels", *Chem. Eng. Sci.*, vol. 56, 2001, pp. 193–210.
10. A. W. Patwardhan, "CFD modeling of jet mixed tanks", *Chem. Eng. Sci.* vol. 57, 2002, pp. 1307–1318.
11. H. D. Zughbi, and M. A. Rakib, "Mixing in a fluid jet agitated tank", *Chem. Eng. Comm.*, vol. 189, 2002, pp. 1038–1056.
12. H. D. Zughbi and M. A. Rakib, "Mixing in a fluid jet agitated tank: effects of jet angle and elevation and number of jets", *Chem. Eng Sci.*, vol. 59, 2004, pp. 829-842.
13. T Raja, P. Kalaichelvi and N. Ananthraman, "Development of CFD model for optimum mixing in jet mixed tanks", *J. Sci. & Ind. Res.*, vol. 66,7, 2007, pp. 5252-527.
14. Y. Swarnalatha, "Studies on Jet Mixing", unpublished.
15. K. SenthilKumar, "CFD studies on Jet Mixing", unpublished.