

Experimental and CFD Investigation of Nozzle Angle in Jet Mixer

Hamid Rafiei, Reza Janamiri, Mohammad Hossein Sedaghat, and Amir Hatampour

Abstract—In this work, the results of mixing study by a jet mixer in a tank have been investigated in the laboratory scale. The tank dimensions are $\frac{H}{D} = 1$ and the jet entrance have been considered in the center of upper surface of tank. RNG-k- ϵ model is used as the turbulent model for the prediction of the pattern of turbulent flow inside the tank. For this purpose, a tank with volume of 110 liter is simulated and it has been divided into 410,000 tetrahedral control cells for performing the calculations. The grids at the vicinity of the nozzle and suction pare are finer to get more accurate results. The experimental results showed that in a vertical jet, the lowest mixing time takes place at 35 degree. In addition, mixing time decreased by increasing the Reynolds number. Furthermore, the CFD simulation predicted the items as well a flow patterns precisely that validates the experiments.

Keywords—Jet mixer, CFD, Turbulent model, Nozzle angle, Mixing time, Reynolds Number.

I. INTRODUCTION

MIXING by jet and impeller are two simplest methods for fluid homogenization in the liquid phase. A jet consists of a pump for fluid circulating, a cheap nozzle and some pipes for transferring the fluid. In mixing process handled by jet, part of liquid is sucked in by a pump and then, will be returned to the tank with a high speed through a nozzle. Injecting fast liquid jet stream into the tank will bound to a great speed within slow layers of the liquid and relatively causes circulation of liquid layers in the tank. After forming the stream by the pump via linking pipes, homogenization liquid and mixing occur. The advantages of jet mixing compared with mixing by impeller are: (1) Jet mixers are cheaper and also easier to install and don't need much resistant construction for the tank while impeller mixers demand a strong engine to circulate and vibrations caused by engine rotation on the tank need, make strong body. (2) This sort has no moving part inside tank, it's easy to keep clean and safe, unlikely, one inside courses corrosion and because of being out of reach in critically practical situations, there's no way but stopping operation to remove defect and costs a lot.

Hamid Rafiei, Faculty of Chemical Engineering, Islamic Azad University, Dashtestan Branch, Iran (phone: +98 937-620-9433; e-mail: rezajanamiri@yahoo.com)

Reza Janamiri, Faculty of Chemical Engineering, Islamic Azad University, Dashtestan Branch, Iran (phone: +98 936-495-8029; e-mail: rezajanamiri@yahoo.com)

Mohammad Hossein Sedaghat, Faculty of Chemical Engineering, Islamic Azad University, Dashtestan Branch, Iran (phone: +98 917-773-7924; e-mail: m.sedaghat66@gmail.com)

Amir Hatampour, Faculty of Chemical Engineering, Islamic Azad University, Dashtestan Branch, Iran (phone: +98 917-772-0395; e-mail: amir.hatampour@gmail.com)

In recent years, based on simulated models by CFD some studies were done on mixing process. Brooker [1] studied jets performance due to CFD and concluded that CFD model can predict mixing time by error of 15%. He studied mixing time and location of probe. In 2001, Jayanti [2] simulated mixing procedure in 2D by using CFD. He studied and assessed stream pattern in cylinder dishes by applying CFX software. He discovered that omitting dead zones in reactor and using conical bottom reduces mixing time. In 2002 Patwadhen [3] simulated jet mixing tank by using CFD which measured mixing time substantially. However, numerical solution and laboratory results of density profile did not have significant adaptation. In 2005, Rahimi and Parvareh [4] compared effect of nozzle situation with suction place in semi-industrial tank by using CFD. They studied the effect of angle between nozzle and suction and also the effect of various turbulence models on mixing time as well as the effect of number of control volume on certainly of results based on RNG-K- ϵ model for huge crude oil tank geometry. They achieved the suitable injection angle for geometry. This research focused on effect of nozzle angle and Reynolds number on mixing time in a tank with the unit dimension in which jet enters through upper side such as tank are located under the ground or anywhere else.

II. EXPERIMENTAL

In these experiments, mixing is examined in a cylinder tank with the diameter of 52cm and the height of 52cm and the volume of 111 liters which is full of water. During the experiments, for studying the process of homogenization, salty water (0.2 molar) is used as a tracer. Fig.1 shows the experimental set up. The procedure is in two steps: in the first step, in a constant mass flow mixing and various nozzle angle is studied and the optimized angle is measured. Then, in the second step, by setting the nozzle angle in a proper direction, mixing time is measured in different mass flows. Thus, in the first step, in each part, after arranging the nozzles angle, pump starts and after a while, 50cc of tracer is injected into the system. According results, as is shown in Fig. 2, mixing time at 35 degree is minimum and by increasing mass flow, mixing time decreases.



Fig. 1 The laboratory mixed tank

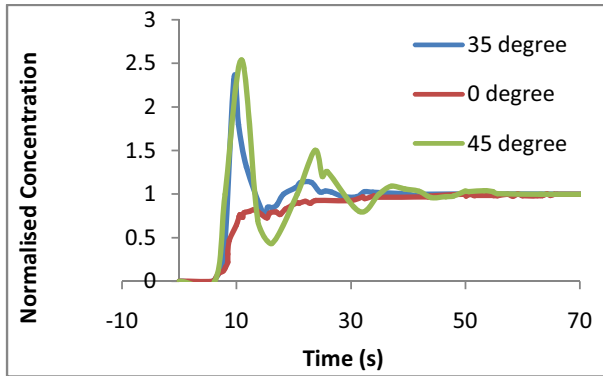


Fig. 2 The experimental mixing curves for various jet setups

III. CFD MODELING

First, geometry of system is designed by GAMBIT software. After making the geometry, the most prominent task is to divide that into smaller parts. Having designed in GAMBIT software, system, the expected geometry is sent to FLUENT to define operating condition and solve Reynolds equation simultaneously. After defining urgent and required speed and primary volume of the system, firstly, mentioned steady equations are solved and after achieving expected convergence, solving the unsteady from of system starts. Equations related to tracer concentration and energy is added to system in addition to solving flow equations and turbulence in unrest position. By having such baffle and also final density, it is feasible to get the eventual time and drawing the textual file explains the tracer density. The chosen fluid is water and operational pressure on a certain spot of the system equals atmospheric plus fluid column pressure on the expected spot.

IV. RESULTS AND DISCUSSIONS

Fig. 3 and Fig. 4 show respectively velocity vector and contours in an axis cut for three angles: 0°, 35° and 45°. As it is shown in Fig. 4, when nozzle is at 35°, two circulation streams are created that cause circulation of fluid in all over the tank that lead to improvement mixing and decrease in mixing time. In Fig. 5, Fig. 6 and Fig. 7 collation of expected CFD and experimental results is evaluated and compared for respectively 0°, 35° and 45° jet. The predicted 95% to 99% overall mixing times for the three jet angle using experimental and simulation models are shown in Table I.

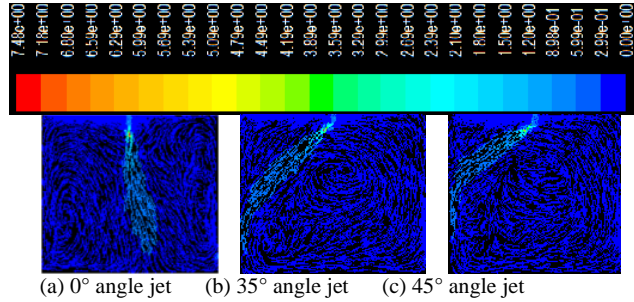


Fig. 3 The velocity vector for different jet angle setups

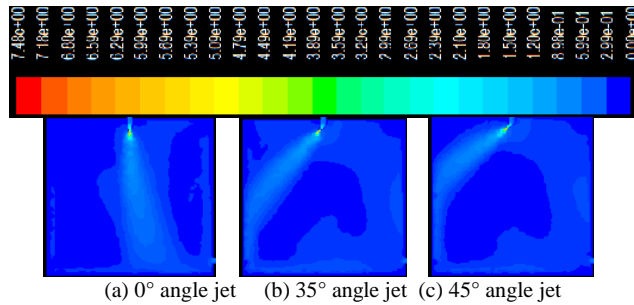


Fig. 4 The velocity contours different jet angle setups

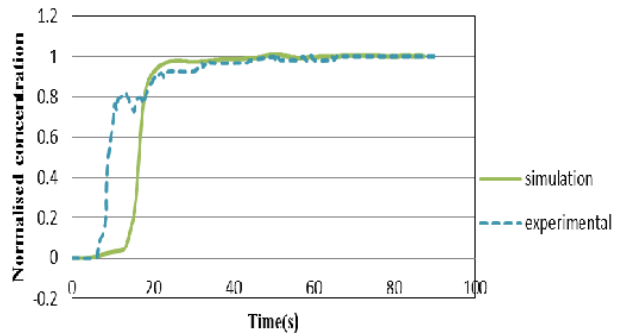


Fig. 5 The comparison between predicted and experimental mixing time curves at 0° jet angle

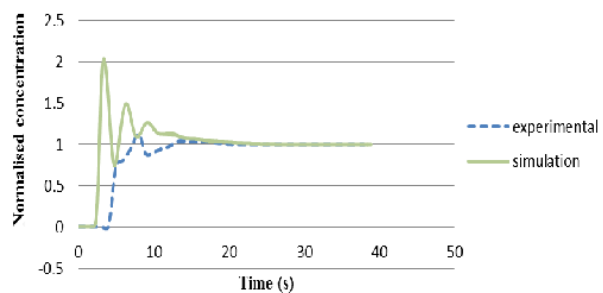


Fig. 6 The comparison between predicted and experimental mixing time curves at 35° jet angle

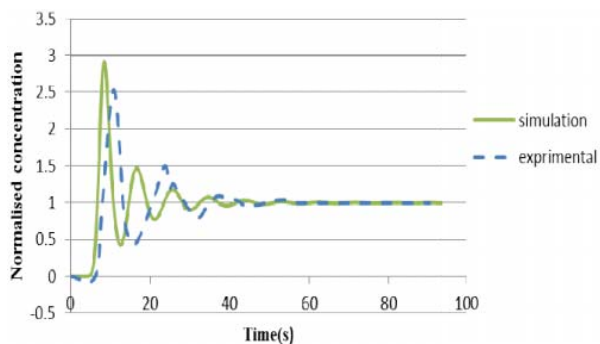


Fig. 7 The comparison between predicted and experimental mixing time curves at 45° jet angle

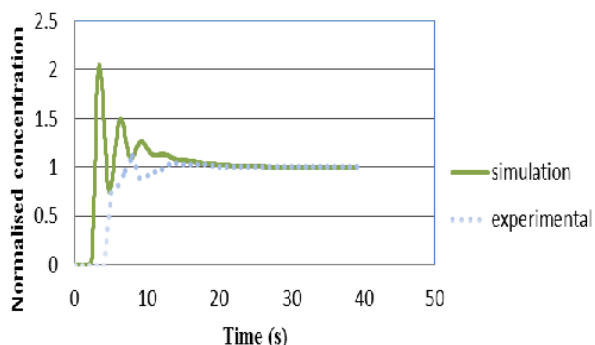


Fig. 8 Predicted concentration profile for 35° angle jet with increase Reynolds

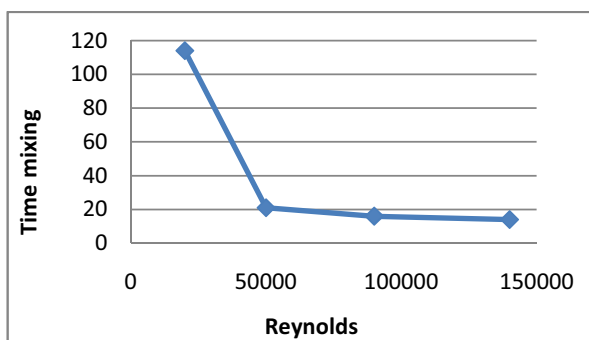


Fig. 9 The effect of number of Reynolds on the mixing time

TABLE I
THE COMPARISON BETWEEN THE EXPERIMENTAL AND SIMULATION MIXING TIME

Jet angle	0°	0°	35°	35°	45°	45°
Mixing Time	95%	99%	95%	99%	95%	99%
Experimental	32	43	14	20	43	47
Simulation	21	42	13	18	36	43

V. CONCLUSION

Based on the results obtained in this study, the following conclusions can be drawn:

1. CFD liquid dynamic calculation is substantially able to predict mixing.
2. Nozzle angle reduces mixing time effectively.

3. By the rise of volumetric flow rate, mixing time falls and any change of the mixing time in low Reynolds is much more notable compared with another one in high Reynolds.

REFERENCES

- [1] Brooker, L., "Mixing with the jet set", Chemical Engineering Science, Vol. 30, No. 16, 1993.
- [2] Jayanti, S., "Hydrodynamics of jet mixing in vessels", Chemical Engineering Science, Vol. 56, pp. 193–210, 2001.
- [3] Patwardhan, A.W., "CFD modeling of jet mixed tanks", Chemical Engineering Science, Vol. 57, pp. 1307–1318, 2002
- [4] Rahimi, Parvareh, A., "CFD study on mixing by coupled jet-impeller mixers in a large crude oil storage tank", Computers and Chemical Engineering, Vol. 31, pp. 737–744, 2006.