Journal of Scientific & Industrial Research
Vol. 66, July 2007, pp. 522-527

Development of CFD model for optimum mixing in jet mixed tanks

T Raja, P Kalaichelvi* and N Anantharaman

Department of Chemical Engineering, National Institute of Technology, Trichy

Received 20 October 2006; revised 13 February 2007; accepted 19 February 2007

A jet-mixed vessel can create a flow pattern similar to that in an agitated vessel with paddle impeller. Tangential flow and circulation flow can be provided by using a jet nozzle fixed at the center of a cylindrical vessel, where circulation flow can be controlled by varying angle of jet nozzle and jet flow rate. Numerical studies were performed using k-ε turbulence models using computational fluid dynamic software FLUENT 6.1 to examine the effects of shape of tank on flow pattern and mixing characteristics in a jet mixer. Results show that better mixing is obtained for cylindrical bottom than the flat bottom for the same total mass flow rate.

Keywords: CFD model, Jet mixed tanks, Mass flow rate

IPC Code: G06D1/00

Introduction

In modern chemical processing units, liquids in a tank are circulated by drawing them through a pump and returning them to the tank through a pipe or nozzle. Setup includes jet injection through a nozzle attached to the sidewall of a cylindrical tank. External recirculation system consists of a pipeline and a pump. Nozzle is directed across a diameter of the tank at mid height of the tank and at an angle inclined to the base. Several researchers[1-3] have proposed the correlating equations of the mixing time. Lehrer[4] formulated a model for free turbulent jet of miscible fluids of different density. Lane & Rice[5] investigated a vertical jet mixer in a vessel with hemispherical base, and observed that the mixing time is strongly dependent on the jet Reynolds number in the laminar regime and only slightly dependent on jet Reynolds number in the turbulent regime. Grenville & Tilton[6] showed that the mixing time is governed by the energy dissipation rate in a region far from the jet nozzle, where the velocities and turbulence intensities are much lower. Unsteady jets[7] have been found more energy efficient than steady jets. Brooker[8] studied the performance of jet mixer using computational fluid dynamics (CFD) and predicted mixing time with an error (15%). Hoffman[9] carried out CFD simulations of the mixing process in a large storage tank using jet. Ranade[10] investigated the flow patterns and mixing in jet mixed tanks equipped with alternating jets using CFD simulations using a standard k-ε model for turbulence but the model was not validated by comparison with the experimental measurements.

This study presents a detailed CFD study to arrive at an optimum geometrical condition of a tank for rapid mixing in the circulation flow regime.

Formulation of Model

CFD model in FLUENT 6.1[11] enables the solution of Reynolds transport equations together with a turbulence model to close the set of equations. In present work, a standard k-ε model was employed to close the Reynolds transport equations, which were discretized by control volume formulation on a staggered grid arrangement. Power law scheme was used for discretization. At the location of nozzle, jet velocity was resolved in the three directions. Values of 'k' will be calculated iteratively depending upon the convective and dispersive transport of 'k' and the gradients of mean velocity that determine the generation and dissipation of 'k'. These simulations are designated as CFD predictions by calculating 'k' iteratively at the jet entry. The mass of tracer used in simulation was same as that used in the experimentation. The effect of mass of tracer would be negligible, since it is a very small fraction of

*Author for correspondence
Tel: 91-431-2501801, Fax: 91-431-250 0133
E-mail: kalai@nitt.edu

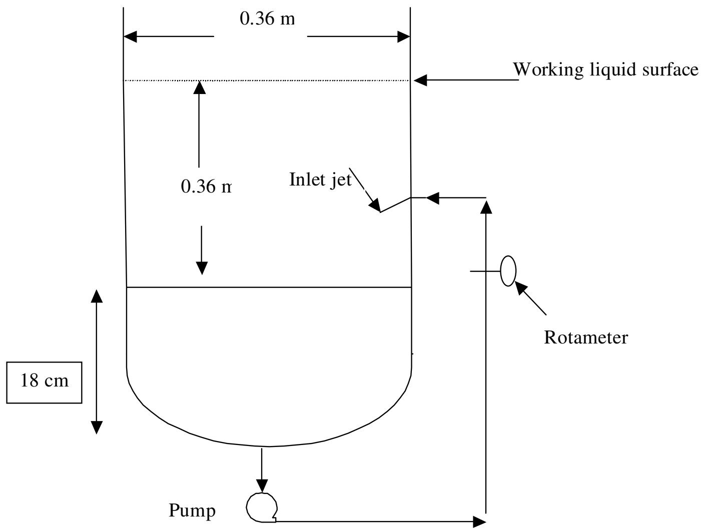

Fig.1— Schematic diagram of a cylindrical tank with cylindrical base

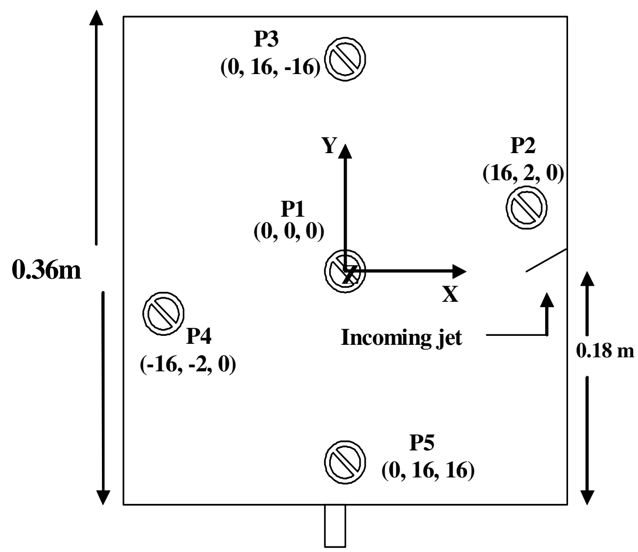

Fig. 2 — Location of conductivity probes (P1 - P5)

the total mass in the tank. Density of tracer was assumed to be the same as that of water since tracer solution was dilute NaCl. Hence, gravity, buoyancy and drag effects on tracer pulse were ignored. These effects need not be considered since tracer pulse had viscosity and density practically same as that of the bulk solution (tap water). In the experiments, tracer input is given with beaker at the center of tank at the top liquid surface. A small amount of turbulence, which is confined to a very small area near surface, is likely to be imparted to the liquid as a result of tracer addition and can be neglected.

Validation Studies

An experiment was set up to change bottom shape of the tank to hemispherical instead of flat (Fig. 1). Various heights of the bottom dish were fabricated and experiments were carried out in a tank (diam, 0.36 m) filled with tap water (height, 0.36 m). NaCl was used as the tracer. Nozzle was inserted at the mid height of the tank. Nozzle inclination[12] of 30° gives a better mixing and hence, in present study, an inclination of 30° was taken.

Conductivity was monitored at four locations using conductivity probe and a chart recorder. Initially, experiment was conducted with 5 probes (Fig. 2). But the reading shown by the probe P1 at the position 0,0,0 did not differ much with the rest of the readings and, therefore, not included in the calculation of the overall mixing time. Mixing time is the time from tracer addition to the time when m= 0.05. The m is calculated as

$$m = \text{abs}[(c-c_e)/c_e)]$$

where, c_e is equilibrium concentration of tracer and c is concentration at a monitoring point at any time. Overall mixing time was considered as the time required for the conductivity to reach within 95% of the fully mixed value. Each mixing time experiment was repeated at least thrice.

Description on Estimation of Mixing Time through Numerical Solution

The velocity field at steady state was obtained for the given geometry, inlet and outlet conditions. This was then given as the initial velocity field for the transient calculations. Initial concentration was set to zero throughout the flow domain except in the inlet, where it was set to a value of 0.2. These conditions were maintained constant throughout the tracer injection period. Evolution of concentration field with the introduction of tracer was then calculated by marching forward in time with a time step of 0.05 sec, which was increased, gradually to a maximum 0.1 sec for large values of time depending on the case being studied. To represent the effect of recirculation and consequent time-varying inlet concentration, the calculations were carried out in short periods (0.5-25 sec) when inlet and outlet concentrations were fixed. At the end of each period, mean outlet concentration added to inlet concentration would be zero. Otherwise, except during the injection period, to represent the effect of feeding back without any time delay, the fluid will be going out through the outlet. The calculations were carried out repeatedly in this manner until the concentration at all points differed (< 0.1%) from the fully mixed value.

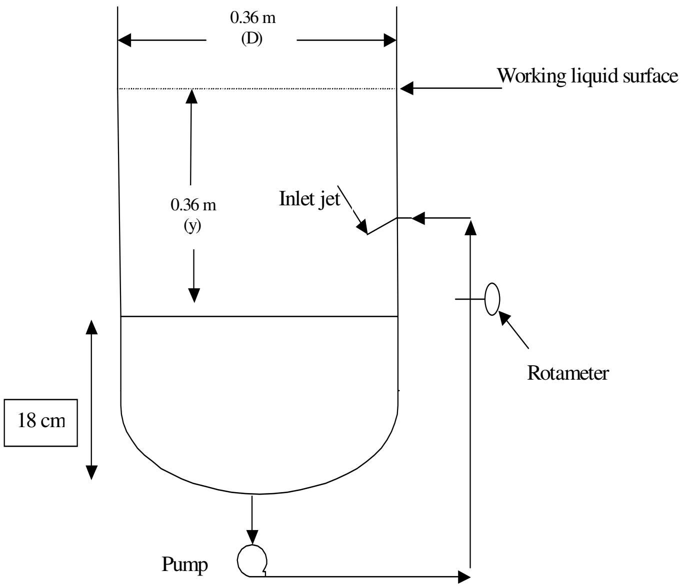

Fig. 3— Schematic diagram of a flat base cylindrical tank

Results and Discussion

Numerical simulation of the mixing process was carried out for four probes (P2 - P5). For all geometry, velocity vectors were plotted in a plane parallel to the issuing jet, where visibility of the flow pattern throughout the tank was excellent.

Flow Pattern in a Cylindrical Tank with a Flat Base

Experiment (Fig. 3) was done on a flat base cylindrical vessel (diam and height, 36 cm each). The properties of fluid are taken that of water. Accuracy of simulation results depends on the number of grids used. The value at every node of the grid was calculated by iteration. Concentration profile was predicted by simulating with 125,000 nodes, which is not the largest, but sufficient. The profiles predicted by higher nodes (216,000) did not differ significantly. Hence for all the concentration profile simulations, only 125,000 nodes were used. Velocity vectors are shown at all points in the flow domain and not only at calculated points.

It was observed that a part of the inlet is short-circuited to the outlet (Fig. 4a). The circulation was better near the wall on the direction opposite to the jet. But

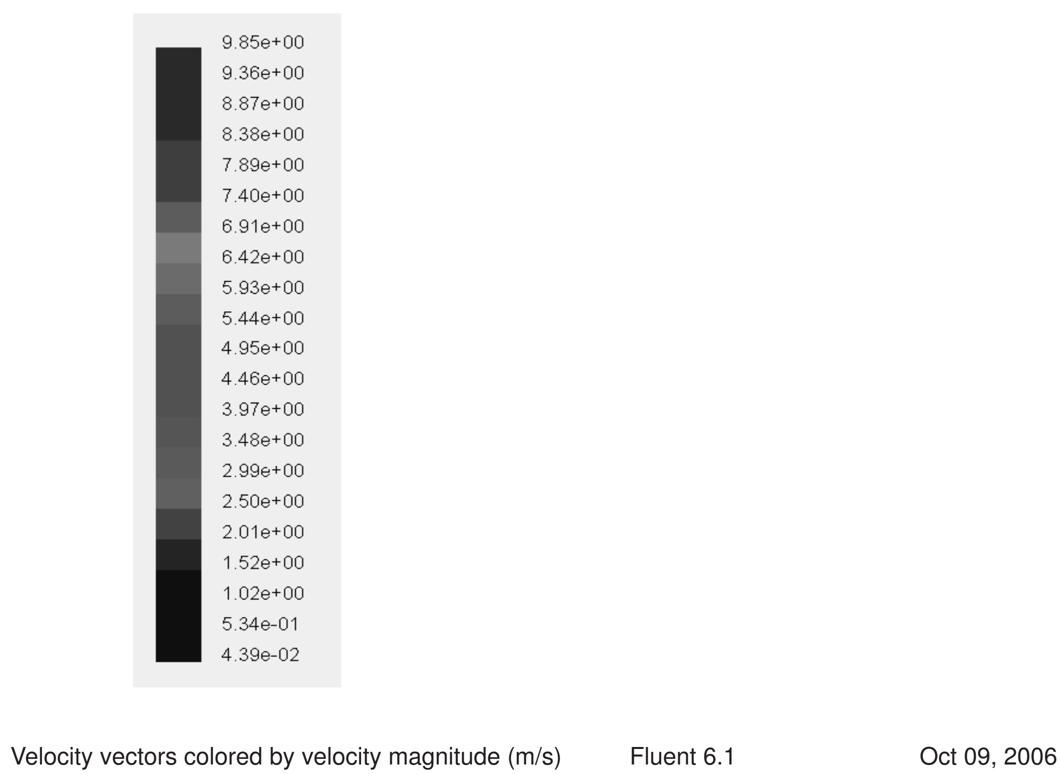

Fig. 4a — Velocity vectors in a plane parallel to the jet for a flat base cylindrical tank