Optimization of Jet Mixer Geometry and Mixing Studies

A. Egedy[1], T. Varga[1], T. Chován[1], B. Molnar[1]

[1]Department of Process Engineering, University of Pannonia, Veszprém, Hungary

Abstract

Mixing is one of the most crucial processes in process industry. The homogeneity in the vessels can be ensured with mechanical, jet or static mixing. Mixing by a jet is one of the simplest methods for fluid homogenization in liquid phase. Jet mixing has become an alternative to mechanical mixing for various applications. The primary aim of use of jet as mixer, like in case of other mixing devices, is to increase the heat and mass transfer between the phases. Beside the injection position the geometry of the jet mixer and the injection nozzle has a major effect on the injection, jet mixing can be effectively examined using CFD simulators. (Patwardhan, 2002, Torré et. al, 2008).With the adequate geometry sufficient mixing can be achieved without using any moving parts. In our study COMSOL Multiphysics® software was used to carry out the investigation of the different jet geometries. The jet mixer was placed in a tube which was used to homogenize the chemical components in a very short reaction zone. The mixing must be complete at the end of the reaction zone because incomplete mixing will lead to certain side reactions. Figure 1.a. shows the modelled geometry with the inlets and the outlet, and the reaction zone, and Figure 1.b. shows and example for the inlet nozzle.

The system has two inlets in this case. A fixed amount of indicator was injected into the inner tube. The indicator induced color changes were recorded, and the results were evaluated using video processing methods. 10 different injection nozzles were applied with different number of inlet tubes. Beside the basic inlet nozzle geometry a swirled type was also created and the effect of the tube angle was investigated. The geometry was implemented in COMSOL Multiphysics®, and momentum and component balances was calculated.

The model was validated against the experimental results using residence time distribution calculation. Besides the inlet nozzle geometry the effect of the flow velocity was also evaluated. Figure 2 shows the streamlines in straight and swirled tube case.
Based on the experimental and numerical investigations an optimal inlet nozzle configuration, and operation parameter range was proposed. The swirled configuration has a better performance than the straight configuration. A MATLAB® GUI was built for the simulation studies, and the program was developed to be a general tool. COMSOL Multiphysics® and MATLAB® were used for the problem formulation, and implementation.

Reference

J.-P. Torré, et. al, „An experimental and CFD study of liquid jet injection into a partially baffled mixing vessel: A contribution to process safety by improving the quenching of runaway reactions, Chemical Engineering Science, 63, 924-942, 2008.
A. W. Patwardhan, CFD modelling of jet mixed tanks. Chemical Engineering Science, 57, 1307-1318

Figures used in the abstract

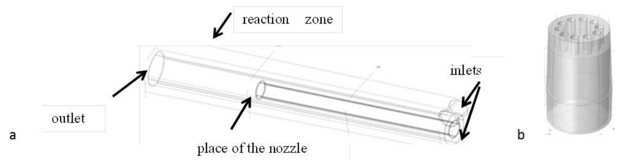

Figure 1: Figure 1.a. An example of the implemented jet mixer, b. An example of the inlet nozzle

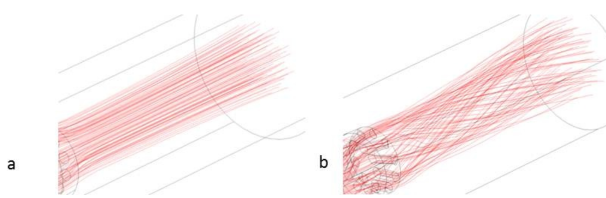

Figure 2: Figure 2 Streamlines in case of a. straight, b .the swirled tubes