335f Computational and Experimental Determination of the Velocity Distribution in a Stirred Reactor with a Retreat Blade Impeller Using Ldv Experimentation and CFD Modeling

Giuseppe Di Benedetto and Piero M. Armenante

Stirred tank reactors equipped with a single retreat-blade impeller and a single baffle are widely used in the pharmaceutical industry to conduct a variety of mixing and reaction/precipitation operations. Most of these reactors, including the vessel, impeller and baffle, are glass lined in order to be resistant to the fluids and the chemicals species used in the reactions and to minimize contamination of the products. Historically, this has significantly limited the ability of equipment manufacturers to fabricate systems with full baffling and optimal impeller configuration. Nevertheless, glass-lined, retreat-blade impeller systems are probably the most common type of reactor configuration used in the pharmaceutical and fine chemicals industry.

Despite their ubiquity, relatively little information is available to date on the hydrodynamics of such retreat-blade impeller systems, although such knowledge can be critical for scale-up, operation, and product quality control, especially if the process is mixing sensitive, as in the case of fast chemical reactions and crystallization.

In this work, the velocity distribution inside a scaled-down, retreat-blade impeller system was experimentally obtained via Laser-Doppler Velocimetry (LDV) and numerically predicted using Computational Fluid Dynamics (CFD). The experimental apparatus consisted of a 30-liter vessel along with a retreat-blade impeller and a single beaver-tail baffle. All three velocity components and the turbulence intensity were measured at different radial positions on a number of horizontal planes inside the vessel. Numerical simulations of the velocity distribution and turbulence levels inside the vessel were conducted using a commercial mesh generator (Gambit) coupled with a computational fluid dynamic (CFD) package (Fluent). The full 360°-tank geometry was incorporated in the simulations.

In general, good agreement between the experimental data and the numerical simulations was obtained. The results reveal a complex flow pattern with strong axial flow near the wall and impeller shaft, and strong radial flow from the impeller itself. The flow was dominated by the tangential component of the velocity as in unbaffled systems, although the presence of the baffle also produced a relatively strong axial flow. The results of this work are expected to be important in process scale-up and optimization, especially if the process is know to be scale-up dependent and mixing sensitive.