

Finding the Right Mix of Computational Fluid Dynamics and Industrial Fluid Mixing Practice in the Design and Analysis of Stirred Tank Reactors

Richard D. LaRoche, Ansys-Fluent, Lebanon, NH, USA

Introduction

It has been over 10 years since the computing and software technologies have existed that allow computational modeling of fluid flow in stirred tank reactors. And yet, there is still uncertainty in industrial practice as to the appropriate use of computational fluid dynamics in the design and analysis of stirred tank reactors. The industrial fluid mixing community has developed sound and reliable engineering methods to model mixing processes in stirred tank reactors. The question is how best to integrate CFD methods with current industrial fluid mixing practice.

How Should CFD Be Deployed?

Within this presentation, the author will review key validations for the use of CFD in modeling stirred tank reactors such as blendtime, velocity fields and impeller power numbers. In the author's view, there is a temptation for those new to CFD modeling to focus blendtime calculations when it is not a particularly useful scale-up parameter.

Also in this presentation, the author will show how CFD can be used as a complementary tool with accepted industrial fluid mixing practice in the following areas.

- Mixing Equipment Design outside standard mixing correlations
- Solid-Liquid Mixing
- Gas-Liquid Mixing
- Liquid-liquid Mixing
- Probability statistics for turbulent energy dissipation and strain rate

The author offers his perspective on how CFD techniques can be used to complement the classic scale-up methods of industrial fluid mixing based on his experience and that of his colleagues while working for Cray Research, DuPont and Fluent.

©Ansys, Inc. 2006