Chemical Engineering Science 61 (2006) 2888–2894

Chemical Engineering Science

www.elsevier.com/locate/ces

CFD analysis of a rotor-stator mixer with viscous fluids

Fabien Barailler, Mourad Heniche, Philippe A. Tanguy*

URPEI, Department of Chemical Engineering, Ecole Polytechnique, P.O Box 6079, Station CV, Montreal, Canada H3C 3A7

Received 1 July 2005; received in revised form 26 September 2005; accepted 4 October 2005
Available online 4 January 2006

Abstract

The characterization of the hydrodynamics of a rotor-stator mixing head has been carried out in the laminar regime with viscous Newtonian fluids. The rotor-stator considered is a very common design composed of a flat blade rotating in a fixed slotted cage. A numerical methodology has been used based on the virtual finite element method to model the velocity patterns, estimate the distribution of shear stress and the flow rate through the head. We have found that the numerical prediction of the power consumption and flow profiles compare well with experimental data. The generation of a pseudo-cavern around the mixing head and how it scales with the Reynolds number have also been investigated, showing that there is a minimum speed limit below which the rotor-stator cannot be used.

Keywords: Rotor-stator; Finite element simulations; Hydrodynamics; Shear; Mixing; Laminar flow

1. Introduction

Rotor-stators are a particular class of mixers commonly used in the petrochemical and cosmetic industries for the production of liquid dispersions and emulsions. When operated at high speed with low-viscosity fluids, they generate a highly turbulent flow field and the pumping discharge is sufficient to maintain the flow in the vessel. Rotor-stators can also be used with viscous fluids. In such a case, the flow around the mixing head is much more complex, exhibiting many of the features associated with non-Newtonian mixing in the laminar regime like segregations, caverns, etc. (Doucet et al., 2005). Although important from an industrial application standpoint, the knowledge of the hydrodynamics in this regime is very limited and deserves more work.

CFD (computational fluid dynamics) is often used to simulate fluid flow with moving parts like in mechanically stirred vessels. This approach provides a wealth of information that can help improving the design of the mixer and its operating efficiency. Detailed velocity, shear rate and flow rate information can also be obtained experimentally with advanced measuring techniques like particle image velocity (PIV) and laser-doppler anemometry (LDA) (Li et al., 2005), but it is essentially a research tool dedicated to small scale bench experiments.

The use of CFD for the flow simulation in a rotor-stator is not an easy task due to the complex topology that continuously changes with time. The virtual finite element method (VFEM), which allows computing flow problems with evolving boundaries without resorting to the re-meshing of the complete flow domain at each time step, will be used in this work. This method is a particular class of fictitious domain method (also called domain embedding method) and it is an advanced version of the Lagrange multiplier fictitious domain method mostly developed by Glowinski et al. (1999, 2000). With the VFEM, the moving parts are not represented as geometrical obstacles which position changes with time but rather by constraining the flow equations in a mathematical sense. The moving parts are in practice replaced by control points on which the velocities are considered as kinematics constraints enforced into the variational formulation of the equations of change by means of Lagrange multipliers. This method was specifically developed for the analysis of flow problems in enclosure containing internal moving bodies. Its application to mixing problems already includes the modeling of a conical helical mixer (Dubois et al., 1996), a helical ribbon mixer (Bertrand et al., 1997), planetary kneaders (Tanguy et al., 2001), twin-screw extruders

* Corresponding author. Tel.: +1 514 340 4017; fax: +1 514 340 4105.
E-mail address: philippe.tanguy@polymtl.ca (P.A. Tanguy).

0009-2509/$ - see front matter
doi:10.1016/j.ces.2005.10.064

(Bertrand et al., 2003) and turbines in centered and eccentric configurations (Rivera et al., 2004).

The numerical modeling literature on rotor-stator hydrodynamics is pretty scarce. Böhm et al. (1998) presented a 2D parallel multigrid finite volume solver to predict unsteady flow in a rotor-stator configuration using a moving-grid technique. They computed the transient phenomena of creation and destruction of eddies during the start-up of a rotor at a Reynolds of 1000. They also highlighted the presence of a secondary flow between the blades of the rotor. Calabrese et al. (2002) performed a 2D sliding mesh simulation of an in-line slotted rotor-stator mixer using the Reynolds average Navier–Stokes (RANS) equations with a k–ε turbulence model closure. He showed that for a turbulent flow at a Reynolds number of 10^4, the mechanical forces generated in the shear gap are not the main responsible for emulsification and dispersion processes. The jets and the swirls discharged from the slots provide the central mechanisms for achieving mixing. Thakur et al. (2002) performed a 3D sliding mesh finite volume simulation of a centrifugal blower using a quasi-steady rotor-stator and k–ε turbulence model. They showed that the prediction of both static pressure rise, horsepower and mass flow rate compared well with experimental data.

In the above studies, the sliding mesh method was used to bypass the excessive CPU time that would be associated with the generation of a grid at each time step. Nevertheless, several issues remain unresolved with the turbulent sliding mesh method in particular the local and global flux conservations on the interfaces (Thakur et al., 2002).

The objective of the present work is the characterization of the hydrodynamics inside and outside the vicinity of a rotor-stator mixing head in the laminar regime with viscous Newtonian fluids. In the forthcoming, we first summarize the experimental part of the study and the hydrodynamics characterization in terms of power draw and flow patterns. Then, we briefly introduce the two computational models used in this work and discuss the simulation results in terms of velocity patterns, shear stress, pseudo-cavern and flow rate through the stator at the light of the available experimental data.

2. Summary of the experimental investigation

A rotor-stator mixer immersed in a 17 L volume vessel was used. It was provided with a shaft mounted torque transducer and a speed encoder allowing to establish the power curve. The rotor-stator head manufactured by VMI-Rayneri (France) is shown in Fig. 1. It is based on a 4-blades impeller rotating in a fixed stator having 72 slotted orifices, 8 aspiration orifices on the upper side and a free opening at the bottom. Fluids tested included Newtonian and non-Newtonian inelastic fluids. The whole setup and experimental results have already been described in Doucet et al. (2005). Their experimental data were used in this work for comparison purposes.

The definition of the Reynolds number for a rotor-stator is ambiguous. One possible definition is that for impellers

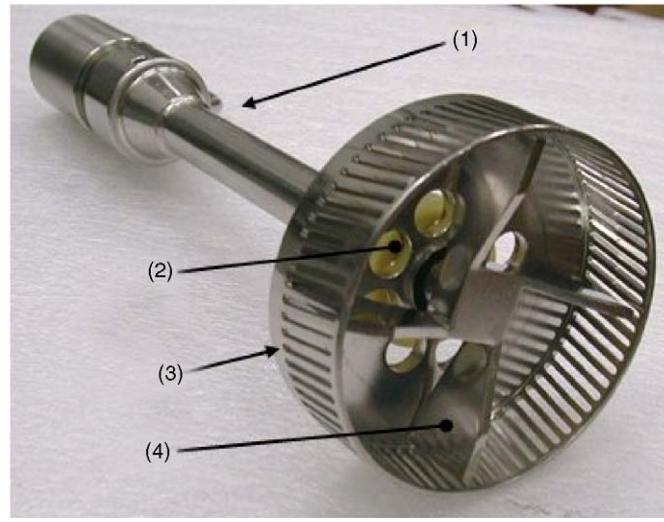

Fig. 1. View of the rotor-stator.

in tanks:

$$Re = \frac{\rho N D^2}{\mu}, \tag{1}$$

where ρ is the density of the fluid, N the rotational speed, D the impeller diameter and μ the viscosity. Another possibility is based on the rotor-stator gap width and the tip speed as characteristics dimensions based on the fact that the power dissipation in the gap would be related to the planar shear. In this case, Re reads as

$$Re = \frac{\rho V_{\mathrm{tip}} \delta_{\mathrm{gap}}}{\mu}, \tag{2}$$

where V_{tip} is the velocity at the blade tip and δ_{gap} the distance between the tip of the blade and the stator.

In the present work, we used the definition provided by Eq. (1) as suggested by Padron (2001) for both experimental and numerical results.

The power curve was derived with Newtonian fluids using the following power number N_P definition:

$$N_P = \frac{P}{\rho N^3 D^5}, \tag{3}$$

where P is the power drawn by the impeller. It has been found that the power constant $K_P = N_P.Re$ is 314 and the power number in the turbulent regime is constant and equal to 3. Glucose solutions having a viscosity from 0.5 to 29 Pa s were used in the experiments. The onset of the transition regime was found to correspond to a Re value of about 100.

An attempt was also made to determine the pseudo-cavern geometry (well-mixed region surrounded by a stagnant fluid) as a function of the hydrodynamics parameters. It appears that the pseudo-cavern size and shape observed with Newtonian fluids scale with the classical definition of the Reynolds number. At low Reynolds numbers, the shape is that of a right circular cylinder. As the Reynolds number increases, the lower part of the well-mixed region grows faster than the upper part reaching