Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2016,28(4):669-675
DOI: 10.1016/S1001-6058(16)60671-6
www.sciencedirect.com/
science/journal/l0016058

Effect of some parameters on the performance of anchor impellers for stirring shear-thinning fluids in a cylindrical vessel*

Houari AMEUR
Institut des Sciences et Technologies, Centre Universitaire Salhi Ahmed, CUN-SA, BP 66, Naama 45000, Algeria,
E-mail: houari_ameur@yahoo.fr

(Received February 10, 2015, Revised June 13, 2015)

Abstract: The 3-D hydrodynamics of shear thinning fluids in a stirred tank with an anchor impeller were numerically simulated. By using a computational fluid dynamics code (CFX 13.0), the obtained results give a good prediction of the hydrodynamics such as the velocity fields and cavern size. The multiple reference frames (MRF) technique was employed to model the rotation of the impellers. The rheology of the fluid was approximated using the Ostwald model. To validate the CFD model, some predicted results were compared with the experimental data and a satisfactory agreement was found. The effects of impeller speed, fluid rheology, and some design parameters on the flow pattern, cavern size and power consumption were explored.

Key words: CFD, computer simulation, stirred tank, anchor impeller, shear thinning fluid

Introduction

Mixing operations with non-Newtonian fluids are frequently employed in areas such as the paint, polymer, food or pharmaceutical industries. Additional difficulties for the optimization of processes often occur with such fluids. Shear thinning fluids are a common class of non-Newtonian fluids, the agitation of such fluids results in the formation of well-mixed zone (known as cavern) around the impeller with essentially stagnant and/or slow moving fluids elsewhere. The formation of the stagnant regions gives rise to poor mass and heat transfer rates, which lead to poor quality of the end products[1]. Thus, the mixing of such fluids is a difficult operation and considered as a key step in the chemical industry. It is desirable to eliminate these stagnant regions by a proper mixing design[2-5].

Low viscosity mixing applications can usually be performed with impeller systems consisting of one or more turbines and propellers. The close-clearance impellers are highly recommended for the mixing of highly viscous fluids, especially for shear thinning fluids, in the laminar regime[6]. For instance, in polymerization reactors, it is desirable to ensure efficient mixing to prevent phenomena like hot spots, to control the molecular weight distribution of the final product, and to avoid the dead zones[7].

Triveni et al.[7] reported that if turbine impellers are used with highly viscous liquids, flow velocities rapidly decay to low values away from the impeller affecting the blending quality. Turbine impellers are therefore not recommended for use in the laminar regime. For such conditions, close-clearance impellers such as anchors are commonly used. Chhabra and Richardson[8] reported that the flow pattern generated by an anchor agitator is tangential and the anchor is suitable for mixing of viscous Newtonian and non-Newtonian fluids. It has been shown that, at higher impeller rotational speeds, an anchor impeller creates secondary axial and radial flows as well[9]. Nagata[10] revealed by experiments that there exists an axial temperature profile within the vessel. Bertrand et al.[11] and Savreux et al.[12] simulated the 2-D laminar mixing of non-Newtonian fluids with an anchor impeller and they confirmed the finding of Nagata that the anchor is inefficient in the laminar regime. Akiti et al.[13] also studied the behavior of an anchor agitated vessel of 2 L and 4 L capacity using CFD and they observed

* **Biography:** Houari AMEUR (1982-), Male, Ph. D., Assistant Professor

that the anchor impeller produces little flow and turbulence in the area beneath the impeller irrespective of the reactor configuration. Karray et al.[14] investigated the performance of the anchor for turbulent Newtonian fluid flow. They found that the use of the classical anchor in turbulent flow yields an important deformation of the anchor arm. To solve this problem, they suggested using an anchor blade. Tanguy et al.[15] measured the power consumption of an anchor agitator for the homogenization of non-Newtonian fluids and they showed that the constant K_s defined by Metzner and Otto[16] do not vary strongly with the power law index (n). Espinosa-Solares et al.[17] studied the combined effect of bottom clearance and wall clearance on the power consumption rate and they proposed a numerical correlation. They have observed that the power consumption decreases as the bottom and wall clearance increase, which is due to the change in the flow pattern.

By experiments, Triveni et al.[18] studied the mixing of Newtonian and non-Newtonian fluids in an anchor-agitated vessel. They observed an increase in the fraction of the well-mixed region from 0.7 to 0.95 with increase in impeller speed for both Newtonian and non-Newtonian fluids but the increase is small for viscous fluids. Anne-Archard et al.[19] studied numerically the hydrodynamics and power consumption in a stirred vessel by helical and anchor agitators. They discussed the Metzner-Otto correlation for yield stress fluids.

By CFD simulations, Prajapati and Ein-Mozaffari[6] investigated the mixing of yield stress fluids for an anchor agitator. They found that the optimum values for the impeller width-to-tank diameter and impeller clearance-to tank diameter ratios were 0.102 and 0.079, respectively. The mixing time and the specific power consumption results for different operating conditions showed that a four-blade anchor impeller is more efficient than a two-blade anchor impeller.

Our search of the literature shows that a little space has been reserved to the prediction of 3-D hydrodynamics of power-law fluids in a tank equipped with an anchor impeller, through CFD modeling. Thus, the purpose of this paper is to simulate the 3-D flow fields generated by an anchor impeller in the agitation of power-law fluids in a cylindrical tank through the CFD technique and to search another design giving better performance.

The effects of fluid rheology, agitator speed, impeller blade width, number of blades and some other design parameters on the flow pattern, cavern size and power consumption were evaluated.

1. Simulated system

Details of the simulated system are shown in Fig.1. It consists of a stirred vessel (diameter: $D =$ 0.3 m, height: $H/D = 1$) fitted with an anchor agitator of 0.006 m×0.012 m blade width which is mounted on a shaft of 0.018 m of diameter (d_s). The liquid level is kept equal to the vessel height. The impeller is placed at a clearance (c) from the vessel base equal to 0.02 m.

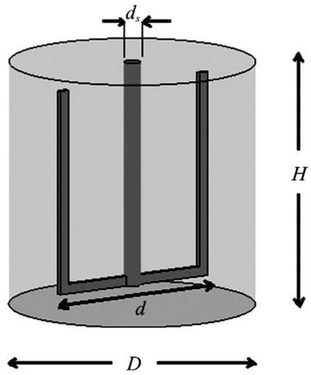

Fig.1 Simulated system

The effect of blade diameter (d) is investigated, four geometrical configurations are realized for this purpose, which are: $d/D = 0.57$, 0.65, 0.73 and 0.82 respectively.

2. Mathematical modeling

The fluid simulated has a shear thinning behavior modeled by the Oswald law. Table 1 resumes the fluid properties (fluid density (ρ), power law index (n) and consistency index (m)) according to the measure of Triveni et al.[7].

Table 1 Properties of the non-Newtonian fluid studied

Fluid	m / kg·[m(s)$^{2-n}$]$^{-1}$	n	ρ / kg·m^{-3}
0.5% CMC	17.78	0.9	1 001.0
1% CMC	8.359	0.4	1 005.5

For non-Newtonian fluids, the apparent viscosity (η) is taken as[2,20]

$$\eta = m(\dot{\gamma}_{avg})^{n-1} \tag{1}$$

The average shear rate is

$$\dot{\gamma}_{avg} = K_s N \tag{2}$$

where K_s is the shear rate constant and N is the impeller rotational speed.

$$K_s = B\left(\frac{3n+1}{4n}\right)^{[n/(n-1)]} \tag{3}$$