Numerical Modelling of Non-Newtonian Fluid in a Rotational Cross-Flow MBR

Bentzen, Thomas Ruby; Ratkovich, Nicolas Rios; Rasmussen, Michael R.; Madsen, S.; Jensen, J. C.; Bak, S. N.

Publication date: 2011

Document Version
Publisher's PDF, also known as Version of record

Link to publication from Aalborg University

Citation for published version (APA):
Numerical modelling of non-Newtonian fluid in a rotational cross-flow MBR

T. R. Bentzen1, N. Ratkovich1, M. R. Rasmussen1, S. Madsen2, J. C. Jensen2 and S. N. Bak1

1Aalborg University, Department of Civil Engineering, Sohngausbroholmvej 57, DK-9000 Aalborg, Denmark. (E-mail: info@civil.aau.dk/terent@civil.aau.dk/jesjensen@grundfos.com)

2Grundfos BioBooster AS, Poul Due Jensens Vej 7, DK-8850 Bjerringbro, Denmark. (E-mail: Bentzen@grundfos.com)

Introduction & Objectives

• Fouling is the main bottleneck of the widespread of MBR systems.
• Process hydraulics can decrease and/or control fouling.
• By increasing liquid cross-flow velocity.
• Rotational cross-flow (RCF) MBR (Grundfos BioBooster®) (Fig. 1)
• Rotating impellers between filtration and aeration membrane discs prevent fouling. It operates up to 5 times higher sludge concentration than in conventional MBR systems (TSS up to 50 g l⁻¹).
• Impellers ensure low viscosity in the reactor biomass due to the non-Newtonian behaviour of acclimated sludge.
• ↓ energy consumption and ↑ flux.

Methodology

Tangential velocity measurements

• RCF MBR operates between 50 to 350 rpm (Fig. 2).
• Experimental tangential velocity measured at 59, 119 and 177 rpm with water.
• Measured with Laser Doppler Anemometry (LDA) (Fig. 3).
• LDA is an optical technique to measure velocity field in transparent media and cannot be used with activated sludge.

CFD model (Fig. 4)

• Star CCM+ V6.02
• Single phase and rigid body motion
• Laminar and k-ε SST

Results and discussion

Tangential velocity measurements

• A good agreement between the experimental measurements and the CFD simulation results, with an error up to 8 % (Fig. 5).

Wall shear stress (Fig. 6)

• It was inferred from CFD simulation that values of the shear stress were accurate (Fig. 7).
• The velocity factor (k) for the RCF MBR was found to be 0.795 ± 0.002 (R² = 0.957), which is within the limits of k for impeller with vanes (0.35 to 0.85).
• CFD model was modified to account for non-Newtonian behaviour for 3 different TSS concentrations (30, 40 and 50 g l⁻¹) and 4 rotational speeds (50, 150, 250 and 350 rpm).
• k was found to be 0.525 ± 0.008 (R² = 0.946), that can be used for the different angular velocities and TSS concentrations.

Area-weighted average shear stress (Fig. 8)

• An empirical relationship, to determine the area-weighted average shear stress in function of angular velocity (in rpm) and TSS was developed:

$$\tau = 3.369 \frac{\eta}{
\dot{V}} = 0.013 \text{TSS}^2 - 2.873$$

Conclusions

• A proper validation of the CFD model was made in terms of tangential velocity measurements using a LDA system with water.
• RCF MBR operates with AS and LDA measurements cannot be made.
• CFD model was modified to account for the viscosity of AS.
• Local shear stress at any place of the membrane surface and area-weighted average shear stress was determined.
• An empirical relationship was made, to determine the area-weighted average shear stress in function of the angular velocity (in rpm) and the TSS.