Advances in Engineering Software 115 (2018) 194–203

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

Research paper

Mixing non-Newtonian flows in anaerobic digesters by impellers and pumped recirculation

Michael Meister[*,a], Massoud Rezavand[b], Christian Ebner[c], Thomas Pümpel[d], Wolfgang Rauch[b]

[a] Department of Environmental, Process, and Energy Engineering, Management Center Innsbruck - The Entrepreneurial School, Innsbruck, Austria
[b] Unit of Environmental Engineering, University of Innsbruck, Innsbruck, Austria
[c] Abwasserverband Zirl und Umgebung, Zirl, Austria
[d] Institute of Microbiology, University of Innsbruck, Innsbruck, Austria

ARTICLE INFO

Keywords:
Anaerobic digester
Computational Fluid Dynamics
Egg-shaped digester
Mechanical mixing
Non-Newtonian flows
Pumped recirculation

ABSTRACT

In this article, a finite volume method based CFD analysis of the mixing of Newtonian and non-Newtonian flows in anaerobic digesters is conducted. The multiple reference frame method is used to model the mixing which is induced by an impeller rotating within a mechanical draft tube. Moreover, the feeding of biomass is considered in the model formulation. Following the validation of the method against reference data, the mixing of cylindrical and egg-shaped digesters is investigated. The distinguishing feature of this article is that the theoretical findings are adopted for the operation of a real-life anaerobic digester. In the context of a case study slurry flows with high total solids concentrations are investigated.

1. Introduction

Greenhouse gas emissions and the resulting global warming necessitate the application of renewable forms of energy as an alternative to fossil fuels. Biogas is a reliable renewable energy source with a very small carbon footprint. Anaerobic digestion (AD) is a widely applied technology for the advanced treatment of biodegradable materials. In AD the biogas, which is formed by anaerobic biological processes from the biological residues, is utilised to generate both heat and energy. A challenge to establish AD as a competitive renewable energy source is to optimise the energy yield [1], whilst at the same time reducing the costs for the operation of the digester.

In order to achieve physical, chemical and biological uniformity within a digester, and to prevent stratification and scum layer formation, the reactor has to be thoroughly mixed [2]. Various mixing techniques with specific advantages, disadvantages and different energy requirements are applicable. Since the energy consumption in AD is dominated by the digester mixing, the optimisation of the energy balance, is subject to ongoing research.

Early experiments on this topic were limited to time consuming full-scale tracer tests [3,4]. Later, the influence of different mixing modes in operational performance was investigated in lab-scale digesters treating cow manure [5,6]. In recent years, Computational Fluid Dynamics (CFD) methods emerged as a promising tool to optimise the mixing in bio-energy systems [7]. Vesvikar and Al-Dahnan [8] performed a steady state CFD simulation of a lab-scale gas-lift digester. They considered an internal airlift reactor, where air is used as the lifting gas. The non-Newtonian fluid flow in a lab-scale mixed-flow digester was simulated in [9]. This study also included simulations of a pilot-scale digester. The lab-scale digester was also studied in [10] using CFD, where it was concluded that the digester mixing speed has no influence on the biogas yield. The biogas yield was maintained at lower velocity gradient values than recommended in the literature. Coughtrie et al. [11] compared different turbulence models for a bench scale gas lift digester. CFD simulations of circular and egg-shaped full-scale digesters were conducted in [12,13]. Karim et al. [14] investigated the effects of different gas-lift digester configurations on the mixing effectiveness. Zhang et al. [15] applied CFD to investigate the mixing mode and power consumption in a stirred tank bio-reactor. An unbaffled reactor was used to study the fluid flow patterns, different biomass compositions and the resulting net power generation.

The aim of this article is an improved operation of the impeller induced mechanical draft tube mixing of cylindrical and egg-shaped digesters by a CFD analysis. Existing methods are enhanced to address important questions which remain unanswered in previous studies. The present improvements of the CFD analysis include a detailed description of the impeller induced agitation of non-Newtonian fluids. Moreover, the feeding of biomass in line with the AD process is

* Corresponding author.
E-mail addresses: michael.meister@mci.edu (M. Meister), massoud.rezavand@uibk.ac.at (M. Rezavand), c.ebner@avzirl.at (C. Ebner), thomas.puempel@uibk.ac.at (T. Pümpel), wolfgang.rauch@uibk.ac.at (W. Rauch).

http://dx.doi.org/10.1016/j.advengsoft.2017.09.015
Received 27 February 2017; Received in revised form 12 September 2017; Accepted 27 September 2017
Available online 05 October 2017

considered. In addition to the mass flow the impact of the feeding pipes on the digester mixing is investigated. The developed model makes it possible to compare impellers' efficiencies for different modes of operation with and without pumped recirculation. The highlight of this article is that the theoretical findings are assembled for a case study to optimise the mixing for the everyday operation of a real-life anaerobic digester.

In the first part of the article, the complete CFD model, which includes the treatment of non-Newtonian slurry flows, is described. In addition, the extensions required for modelling the impeller induced mixing of anaerobic digesters, e.g. the Multiple Reference Frame (MRF) method, are specified. The second part is devoted to the results section, which includes a validation of the model and a mesh refinement study. Subsequently, a mixing analysis with a comparison of the efficiency of different impellers is conducted for a cylindrical digester. As a next step, the previous findings are adopted for a mixing analysis of a real-life egg-shaped anaerobic digester. In this case study, the implications of the pumped recirculation on the flow pattern are investigated. Finally, the effects of different total solids concentrations (TS, expressed in %) on the digester mixing are examined.

2. Methods

In this paper, ANSYS Fluent version 16.2 [16] is employed for solving for an isothermal fluid, which is enclosed in a digester, with the Finite Volume Method (FVM). The FVM is suitable for this problem, because of its conservation properties, the consistency of the method and the support of unstructured meshes. For further details the reader is referred to a standard FVM textbook like [17].

The slurry manure within the digester can be approximated as a single phase flow of constant density due to the high level of mixing being present in this application. Newtonian ($TS = 0\%$) and non-Newtonian slurry flows with different *TS* concentrations in the range of 2.5% to 12.1% are modelled to study the effects of the *TS* concentration on the digester mixing. The physical properties of the slurry manure at a constant temperature of 35 °C, according to the characterisation by Eshtiaghi et al. [18], are summarised in Table 1. In CFD simulations of slurry flows, care must be taken with the selection of rheological characteristics of the sludge, since its properties can differ depending on its composition (see e.g. [19] for a comparison between different sludges). For example, in [20] it is shown that the digested sludge is a shear-thinning yield stress fluid which has different behaviours at varying shear rates.

The slurry flow within the digester is mixed by an impeller which is rotating with a high agitation speed within a draft tube. In the FVM, the impeller induced mixing can be modelled by the MRF method [21]. The basic principle of the MRF method is to divide the simulation domain into regions with different translational or rotational velocities. In each frame absolute velocity components are calculated and continuity is enforced at the interface of the two frames. For a digester which is mixed by a single impeller, the rotating reference frame (RRF) is centred on the mixing device. The rotational speed of this zone is set equal to the agitation speed of the impeller. The remaining parts of the simulation domain conform to a stationary zone where the governing equations are solved in its stationary form. It is recommended that the height of the RRF exceeds 1.5 times the impeller's height [13] such that no unphysical discontinuities arise at the interface. A similar ratio is desirable for the impeller's diameter. However, herein this rule cannot be complied with, because the size of the RRF is limited by the space in between the impeller and the draft tube. Therefore, the largest possible diameters are specified for each scenario. The height and diameter of the RRF for the impeller of type "4PBT45" are $3{\cdot}10^{-1}$ m and $5{\cdot}10^{-1}$ m respectively. In the case study, the RRF measures 1.1 m × $3.45{\cdot}10^{-1}$ m.

Table 1
Physical properties of slurry flows with different *TS* concentrations (Original data of [22,23]).

TS (%)	ρ (kg m^{-3})	min(η) − max(η) (Pa s)	k (Pa s^n)	$\dot{\gamma}$ (s^{-1})	n (10^{-1})
0	998	Newtonian	Newtonian	Newtonian	10
2.5	1000.36	6–$8{\cdot}10^{-3}$	$4.2{\cdot}10^{-2}$	226–702	7.1
5.4	1000.78	1–$3{\cdot}10^{-2}$	$1.92{\cdot}10^{-1}$	50–702	5.62
7.5	1001	$3{\cdot}10^{-2}$–$1.7{\cdot}10^{-1}$	$5.25{\cdot}10^{-1}$	11–399	5.33
9.1	1001.31	$7{\cdot}10^{-2}$–$2.9{\cdot}10^{-1}$	1.052	11–156	4.67
12.1	1001.73	$2.5{\cdot}10^{-1}$–2.93	5.885	3–149	3.67

2.1. Governing equations

2.1.1. Continuity and momentum equation

Firstly, the continuity equation ensures that the velocity field is divergence free everywhere:

$$\nabla{\cdot}\boldsymbol{v} = 0 \tag{1}$$

The variables ρ and $\boldsymbol{v}$ define the density and absolute velocity vector of the fluid. Secondly, the rate of change of velocity is controlled by the momentum equation:

$$\frac{\partial}{\partial t}(\rho\boldsymbol{v}) = -\nabla{\cdot}(\rho\boldsymbol{v}\boldsymbol{v}) - \nabla p + \nabla{\cdot}\boldsymbol{\tau} + \boldsymbol{F} \tag{2}$$

The variables p, $\boldsymbol{\tau}$ and $\boldsymbol{F}$ denote the fluid's pressure, viscous stress tensor and the body forces.

The equations above are expressed in terms of the absolute velocity vector. However, in proximity to the mixing device a reference frame which is rotating with the angular velocity of the impeller is specified by using the MRF method. The relationship between the absolute and relative velocity vector $\boldsymbol{v}_r$ in the RRF is given by:

$$\boldsymbol{v} = \boldsymbol{v}_r + \boldsymbol{\omega} \times \boldsymbol{r} \tag{3}$$

In Fluent, the velocities in each sub-domain are calculated in the frame of motion of the corresponding sub-domain. At the interface between two sub-domains continuity of the absolute velocity vector is enforced.

Therefore, in the RRF the momentum equation is complemented by a term describing the Coriolis and the centripetal accelerations. In presence of gravitational acceleration $\boldsymbol{g}$ this yields

$$\frac{\partial}{\partial t}(\rho\boldsymbol{v}) = -\nabla{\cdot}(\rho\boldsymbol{v}_r\boldsymbol{v}) - \rho(\boldsymbol{\omega} \times \boldsymbol{v}) - \nabla p + \nabla{\cdot}\boldsymbol{\tau} + \boldsymbol{g}. \tag{4}$$

2.1.2. Non-Newtonian fluid model

Slurry flows with $TS \geq 2.5\%$ can be described as non-Newtonian pseudo-plastic fluids. Different mathematical formulations are found in the literature to model non-Newtonian fluids [24]. In the power-law model the dynamic viscosity η is computed from the shear rate $\dot{\gamma}$ [26]:

$$\eta = k\,\dot{\gamma}^{n-1} \tag{5}$$

The variables k and n denote the consistency and the power-law index. Note that the power-law index is restricted to $n < 1$ for pseudo-plastic fluids. The power-law model is only applicable to a limited range of shear rates, because the limits of zero and infinite shear viscosities are not supported.

An alternative model which supports these viscosity limits is the formulation by Carreau [27]:

$$\frac{\eta - \eta_\infty}{\eta_0 - \eta_\infty} = (1 + \lambda^2\dot{\gamma}^2)^{(n-1)/2} \tag{6}$$

where η_0 and η_∞ denote zero and infinity shear viscosities. The variables γ and n are curve-fitting parameters.

A suitable model for low shear rates is the Ellis model [24]. In simple shear the apparent viscosity of the fluid can be obtained by the following equation: