Chemical Engineering Journal 298 (2016) 259–270

Contents lists available at ScienceDirect

Chemical Engineering Journal

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

Fluid dynamic analysis of non-Newtonian flow behavior of municipal sludge simulant in anaerobic digesters using submerged, recirculating jets

Baharak Sajjadi [a], Abdul Aziz Abdul Raman [a,*], Rajarathinam Parthasarathy [b]

[a] Department of Chemical Engineering, Faculty of Engineering, University of Malaya, 50603 Kuala Lumpur, Malaysia
[b] School of Civil, Environmental and Chemical Engineering, RMIT University, Victoria 3001, Australia

HIGHLIGHTS

- Unsteady jet mixing of non-Newtonian flow in an anaerobic digester was analyzed.
- The study was conducted by integrating of computational and experimental analysis.
- Effects of rheological properties and input power on cavern were analyzed.
- Cavern was analyzed in terms of its formation, growth, structure and volume.

ARTICLE INFO

Article history:
Received 27 November 2015
Received in revised form 9 March 2016
Accepted 15 March 2016
Available online 2 April 2016

Keywords:
Anaerobic digestion
Non-Newtonian flow
Sludge rheology
Submerged jet
Computational Fluid Dynamics (CFD)

ABSTRACT

The current study aims at analyzing unsteady jet mixing of non-Newtonian fluids and the cavern structure in order to reduce the volume of stagnant zone in anaerobic digester tanks. The study was conducted by integrating computational analysis of the system into the experimental analysis. Accordingly, a highly viscous xanthan gum solution that mimics sludge with high solids concentrations was used to simulate municipal wastewater. Mixing was provided by a submerged, downwards-pointing and recirculating jet. Besides, a Computational Fluid Dynamics (CFD) model was developed to investigate different flow regimes, cavern formations, cavern growths, cavern volumes and the effects of rheological properties and specific power input on them. The variation in the xanthan gum rheology due to the digestion process was considered using the power law model. CFD was accomplished to investigate the hidden details of the system and experimental tests were used for validating CFD results. It was found that the bilaterally symmetrical structure of the cavern depended on the rheological properties and position of the outlet point, which directly affected the volume fraction of active/inactive zones. Accordingly, three different flow regimes for the cavern structure and the reduction trend of the inactive zone were recognized. Comparison of three different power inputs also demonstrated that reduction of inactive zones was not significantly affected by the power input beyond the optimum power input. Another key parameter was found to be the position of the outlet. It was found that active materials made more than half to 80% of the outlet while only half of the digester was filled with active materials.

1. Introduction

Currently, over 50% of the world's water supplies (rivers, lakes, coastal waters) are severely polluted with untreated domestic, industrial and agricultural wastewater. According to United Nations Environment Programme (UNEP), over half of the world's population will face water shortages in the next 30 years [1]. Accordingly, wastewater treatment is of concern in most developed and developing countries. Anaerobic digestion is a biological and economical process for treating organic wastes, recovering energy from waste products and diminishing excess biological sludge from wastewater treatment plants. Moreover, the by-products of anaerobic digestion (i.e. carbon dioxide and methane) can be used for supplying electricity and heat to the process. However, the efficiency of this process is critically dependent on sludge rheological properties and mixing hydrodynamics in digesters.

* Corresponding author. Tel.: +60 3 79675300; fax: +60 3 79675319.
E-mail address: azizraman@um.edu.my (A.A.A. Raman).

http://dx.doi.org/10.1016/j.cej.2016.03.069
1385-8947/

Nomenclature

c	Geometry constant value [−]	N	Number of pulse jet mixers [−]
P	Driving pressure [Pa]	P	Power input [W]
T	Temperature [K]	$\dot{Q}$	Liquid recirculation rate [m^3/s]
u	Liquid velocity [m/s]	u_d	Injection velocity at the nozzle [m/s]
k	Turbulent kinetic energy [m^2/s^2]	V_i	Inactive volume [m^3]
ε	Turbulent dissipation rate [m^2/s^3]	V_T	Total volume of the vessel [m^3]
μ_t	Turbulent viscosity [Pa.s]	V_a	Active mixing volume [m^3]
μ	Dynamic viscosity [Pa.s]	SST	Shear stress transport [−]
ρ	Fluid density [kg/m^3]	τ	Shear stress [Pa]
σ	Surface tension [N/m]	$\dot{\gamma}$	Strain rate [1/s]
α	Volume fraction [−]	τ_y	Yield shear stress [Pa]
d_s	Suction point diameter [mm]	$\dot{\gamma}_c$	Critical shear rate [s^{-1}]
d_N	Injection nozzle diameter [mm]	H	Height [m]
d_0	Nozzle diameter [cm]	D	Diameter [m]
h	Distance between the injection point and tank bottom [m]	L	Liquid [−]
		C	Cavern [−]
N_t	Normalized time-scale [$N_t = \dot{Q}t/V_T$]	T	Tank [−]

Currently, three types of mixing that include mechanical mixing, gas mixing and jet mixing are employed in industrial digesters. The comparison of these methods in anaerobic digesters has been well documented in the available literature [2–4]. The majority of the published research focuses on fluid flow characteristics caused by mechanical mixing or gas mixing. On the other hand, rheological behaviors of sludge play a key role in sludge treatment and management. Most of the input materials for anaerobic digesters (i.e. wastewater, slurries from food processing plants and animal manure) often exhibit non-Newtonian rheology. As opposed to Newtonian fluids in which viscosity is generally constant and the shear stress is directly proportional to the shear rate, this linear relation is not valid in non-Newtonian fluids. Non-Newtonian flow in a digester may lead to non-uniform and non-ideal flow conditions, incomplete mixing, short circuiting and increment of inactive and stagnant zones. An extensive review of non-Newtonian fluid flow in anaerobic digesters was conducted by Wu and Chen [5]. The authors then developed a mathematical model integrating theories of non-Newtonian fluid and single-phase turbulence in mixed-flow anaerobic digesters and depicted that the flow pattern for non-Newtonian fluid completely differed to that of the Newtonian.

Apart from experimental investigation, Computational Fluid Dynamics (CFD) has been established as a trustable technique for numerical analysis of such systems. CFD technique provides valuable information about a system (herein digester) such as its hydrodynamic performance even before the construction of experimental setup. CFD can also clarify the hidden key factors of the mixing phenomena in a digester. Literatures on CFD simulation of different aspects of mixing in anaerobic digesters were reviewed in order to fulfill the current study's objectives. The results were categorized according to mixing methods and summarized in Table 1. In general, the theories of jet mixing in non-Newtonian fluids are developing. Therefore, there are limited studies in using CFD simulation in this field. Besides, the majority of the published research (as listed in Table 1) for non-Newtonian fluids is associated with mechanical agitators.

Mixing of non-Newtonian fluids with impeller, mixing jet or gas sparger leads to the formation of an enclosed and well-mixed zone around the mixing agent named "Cavern" surrounded by a stagnant zone. The cavern boundary can be defined as the surface where the fluid's yield stress equals the local shear stress. Fluid velocity at some points of the non-Newtonian flow is so low that fluid stresses are no longer able to overcome the shear. Therefore, a stable force balance happens and the flow is stopped at the cavern boundary. Analyzing the physics of the cavern is one of the most complex problems encountered in non-Newtonian fluid flows. Several researchers have tried to correlate cavern formation and its size to the power drawn by the yield stress fluid. These models are collected in Table 2. The models for different cavern shapes, considering axial and tangential forces, different flow regimes inside the cavern and different rheological models are shown. However, very few theoretical models that combine cavern formation and turbulent mixing jets in non-Newtonian fluid flows can be found. Meyer and Etchells [6] presented two theoretical models for the cavern formed from a single, downward pointing jet in a non-Newtonian fluid. In the Meyer and Etchells' models, cavern size is related to the flow and physical properties of an intermittent jet. The dimensional analysis of the models suggested yields Reynolds number, jet Reynolds number, normalized pulse time and the ratio of yield stress to shear strength as the four most important parameters that affect cavern size and mixing time. The models presented by Meyer and Etchells, can also be employed in multi-jet mixing. The primary difference is that the nozzle diameter should be substituted by the effective nozzle diameter ($d_{0e} = c\sqrt{N}d_0$). The authors have also depicted that a strong central upwelling flow in multi-pulse jet mixings is generated which causes the flow to move radially inward and then turns up at the center of the tank, leading to breakthrough at the surface and leaving an annulus of stationary slurry. In such a situation, cavern behavior cannot be predicted by using the basic model for cavern height as increment in jet velocity may increase the diameter of the breakthrough region. The existence of stagnant regions may also lead to ineffective mass and heat transfer, which directly affects the quality of degradation efficiency or production loss. Eradication of such undesired stagnant zones is always beneficial to increase the process efficiency in the mixing of non-Newtonian fluids.

Accordingly, the current study aims at investigating the mixing characteristics of non-Newtonian flow induced by submerged and recirculating jets to provide a detailed insight into the formation of cavern, its volume, growth and structure, different flow regimes in an anaerobic di satisfactorily gester and the effects of rheological properties and specific power input on these parameters. This study attempted to find a link between the specific power input and rheological properties of a fluid with cavern formation as it