Optimization the Mixing in a Gas-lift Anaerobic Digester of Municipal Wastewater Sludge

D. Baveli Bahmaei[1], Y. Ajabshirchi[2], Sh. Abdollahpour*[3], S. Abdanan Mehdizadeh[4]

1. Ph.D student , Department of Biosystems Engineering, Collage of Agricultural Department, University of Tabriz, Tabriz, Iran.
2. Professor, Department of Biosystems Engineering, Collage of Agricultural Department, University of Tabriz, Tabriz, Iran
3. Associate professor, Department of Biosystems Engineering, Collage of Agricultural Department, University of Tabriz, Tabriz, Iran
4. Assistant professor, Department of Mechanics of Biosystems Engineering, Collage of Agricultural Engineering and Rural Department, Agricultural Sciences and Natural Resources University of Khuzestan, Ahvaz, Iran

(*- Corresponding Author Email: shams@tabrizu.ac.ir)

Abstract

This research aimed to optimize mixing in gas-lift anaerobic digesters of municipal sewage sludge as uniform mixing is important for effective communication between methanogenic bacteria and nutrients. Wastewater municipal sludge sampling was performed at the Ahvaz West treatment plant (Chonibeh) during the summer of 2022. A model was implemented to simulate, optimize and confirm the simulation process using Computational Fluid Dynamics (CFD) by ANSYS Fluent software 19.0. The inlet-gas velocity to the digester was determined, and a draft tube and the conical hanging baffle were added to the digester design. Different inlet-gas velocities were investigated to optimize mixing in the digester, and evaluation indexes such as the sludge particle velocity, the gradient of sludge particle velocity, turbulence kinetic energy, and eddy viscosity of sludge particles were evaluated. The optimal inlet-gas velocity was determined to be 0.3 ms^{-1}. The simulation results were validated using the Particle Image Velocimetry (PIV) method, and there was the sufficient percentage of correlation between the CFD and PIV contours (98.8 % at the junction of the wall to the bottom). The results showed that the model used for simulating, optimizing, and verifying the simulation process was successful. It can be recommended for gas-lift anaerobic digesters, which consist of a cylindrical tank with a height-to-diameter ratio of 1.5, draft

tube-to-digester diameter ratio of 0.2, draft tube-to- fluid height ratio of 0.75, the conical hanging baffle distance from the fluid level equal to 0.125 times the fluid height, and its outer diameter-to-digester diameter of 2/3.

Keywords: Computational Fluid Dynamic (CFD), Digestion, Particles Image Velocimetry (PIV), Simulation.

Introduction

The performance of anaerobic digester is affected by several factors, including the retention time of the substrate within the digester and the degree of contact between the incoming substrate and the viable bacterial population. These parameters are determined by the flow pattern, or mixing, within the digester. Complete mixing of the substrate within the digester facilitates the uniform distribution of organisms and heat transfer. This is considered to be essential in high-rate anaerobic digesters (Sawyer and Grumbling, 1960 and Meynell, 1976).

Three methods of mixing in anaerobic digesters include gas mixing, mechanical mixing, and pumped recirculation liquid. Gas mixing can be performed using either unconfined or confined methods. In unconfined systems, biogas collected at the top of the digester is compressed and discharged through bottom diffusers or top-mounted lances. (McFarland, 2001). In order to make the four gas mixing designs (Bottom diffusers, Gas lift, Cover mounted lances, and Bubble guns) comparable, MEL = 5 Wm^{-3} at TS = 5.4% was used to determine the gas inlet velocity. In confined systems, the biogas is released through tubes. The gas lift, a confined system, produces the highest average velocity (0.080 ms^{-1}) under the same mixing power (5 Wm^{-3}). In other words, mixing with the gas lift requires the lowest mixing power under the same average velocity of the flow field, and is therefore recommended (Wu, 2014).

The flow pattern, or mixing, inside gas-mixed digesters is affected by several factors, including the biogas recycling rate, the bottom clearance of the draft tube, the ratio of the draft tube to tank diameter, the slope of the hopper bottom, the position and design of the biogas injection (sparger), and the solids loading rate, among others (Karim *et al*., 2005). Wei *et al*. (2023) assessed the impact of treated sludge rheology as an important factor on flow and mixing characterization in their study on optimizing flow and mixing in a full-scale biogas-mixed digester.

Experimental experiments to evaluate the effect of all these parameters on mixing within the digester are time-consuming and costly, so simulation software such as ANSYS Fluent are suitable tools for designing and optimizing gas mixed anaerobic digester. Wu (2010) presented a Eulerian multiphase flow model to solve gas mixing in digesters, and proposed that the shear stress transport (SST) k–w model with Low-Reynolds corrections could be an appropriate turbulence model to solve gas and non-Newtonian two-phase flow.

Researchers use different indexes to determine the performance of their simulations and to be able to evaluate simulations performed with experimental data. Varma and Al-Dahhan (2007) measured velocity and turbulence kinetic energy. Karim *et al*. (2007) measured magnitude axial velocity. Wu (2010) determine the velocity counter, Wu (2014) used the average velocity and uniformity index for velocity to evaluate the mixing performance, and Daplo *et al.* (2015) used velocity magnitude along a vertical axis.

Validate the CFD simulation results is necessary. Tracer and non-invasive techniques are traditional methods for studying gas mixing in anaerobic digesters. These methods usually are used to verify the CFD simulation results. Vesvikar and Al-Dahhan (2016), Karim *et al*. (2007), and Wu (2010) validated their models by digester reported by Karim *et al*. (2004) and verified the flow fields with the measured data from Computer Automated Radioactive Particle Tracking (CARPT) and Computed Tomography (CT) (non-invasive technique). Dapelo *et al*. (2015) used Particle image velocimetry and high-speed camera for validated Euler-Lagrange CFD model of unconfined gas mixing in anaerobic digestion. Hu *et al.* (2021) proposed a novel approach for experimentally quantifying the mass transfer in High solid anaerobic digestion's mixing process in a mixing tank equipped with multistage impellers by means of the Laser Induced Fluorescence (LIF) technique. Flow field was investigated for better illustrating the mass transfer, thus Particle Image Velocimetry (PIV) and Computational Fluid Dynamics (CFD) technique were conducted for flow field measurement.

The quality of mixing in a gas-lift anaerobic digester depends on various factors, such as the dimensions of the draft tube and the conical hanging baffle, the position of the baffle relative to the digester bottom, and the angle of the baffle. Baveli Bahmaei *et al.* (2022) performed a numerical study to examine the influence of these factors on the mixing performance using ANSYS Fluent software. The present paper extends their work by optimizing the mixing in the same digester configuration with different inlet-gas velocities. The evaluation criteria for the optimization are average velocity, turbulence kinetic energy, average velocity gradient, and sludge eddy viscosity. The numerical results are validated by particle image velocimetry (PIV) measurements.

Material and method

Methodology

The Computational Fluid Dynamics (CFD) simulations were conducted using ANSYS Fluent software to model the inlet-gas anaerobic digester. The initial step involved determining the inlet-gas velocity to the digester. Subsequently, the effects of adding the draft tube and the conical hanging baffle to the digester design were investigated. The optimization of mixing within digester was achieved by varying the inlet-gas velocities and assessing the sludge particle velocity, gradient of sludge particle velocity, turbulence kinetic energy, and eddy viscosity of sludge particles as evaluation indexes. The resulting evaluation index contours were analyzed to determine the optimal velocity for mixing. Following the simulation results, a transparent anaerobic digester was constructed and loaded with municipal sewage sludge, operating at optimal inlet-gas velocity. The Particle Image Velocimetry (PIV) method was employed to validate the CFD simulation outcomes by comparing the evaluated index contours of PIV with those of CFD. A schematic representation of the simulation, optimization, and verification process is presented in Fig. 1.

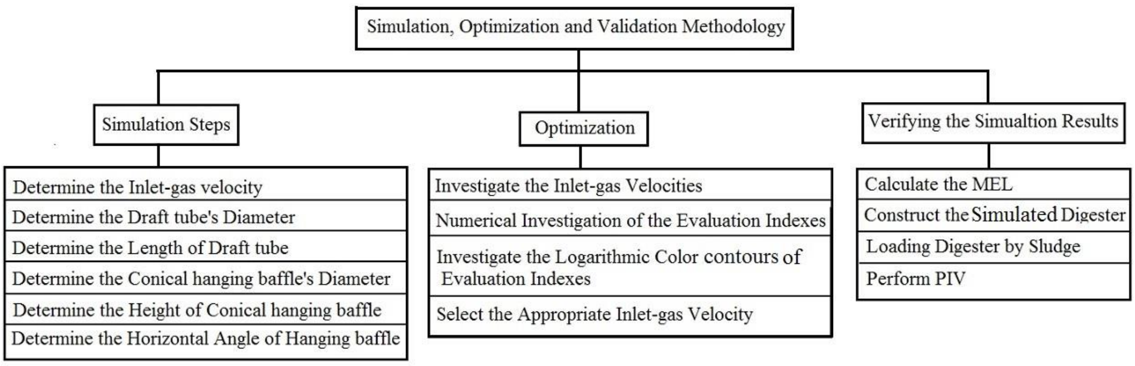

Fig. 1: The model of simulation, optimization and verification of the process

CFD simulation

The commercial CFD software ANSYS Fluent 19.0 was utilized to create two-dimensional geometry in the design modeler, generate a mesh and solve the two-phase

Eulerian model flow using the Eulerian multiphase approach. This two-dimensional model can be applied to model digesters with an axial symmetry structure (Yang *et al*., 2015). Simulations were performed under unsteady-state conditions, using Double Precision, Serial, Pressure-Based and Implicit settings. The two-phase liquid-gas Eulerian Model of Viscous-SST k-omega (with sludge as the primary phase and biogas as the secondary phase) and low-Re correction were employed. At each time step, the iterative calculation was considered converged when all residuals fell below 1×10^{-3}. Final convergence was achieved when the average velocity of liquid phase remained unchanged (Wu, 2014).

Geometry, Computational domain and mesh

The digester geometry used in this research was based on the simulated digester geometry outlined in the six steps of digester simulation mentioned in Fig. 1 by Baveli Bahmaei *et al*. (2022). The digester consisted of a cylindrical tank with a flat bottom and a height to diameter ratio of 1.5 (45 to 30 cm). The draft tube diameter to digester diameter was 0.2 (5 cm) and the draft tube height to fluid height was 0.75 (30 cm). The conical hanging baffle distance from the fluid level was equal to 0.125 of the fluid height (5 cm), its outer diameter to digester diameter was 2/3 (20 cm), and a horizontal angle of 15 degrees (Fig. 2). For simulations, the Mesh Size Function was set to Curative, Max Face Size was set to 0.0007, and the number of Nodes and Elements were 267083 and 264281, respectively. Discretization error estimation was conducted according to the method proposed by Celik *et al*. (2008).

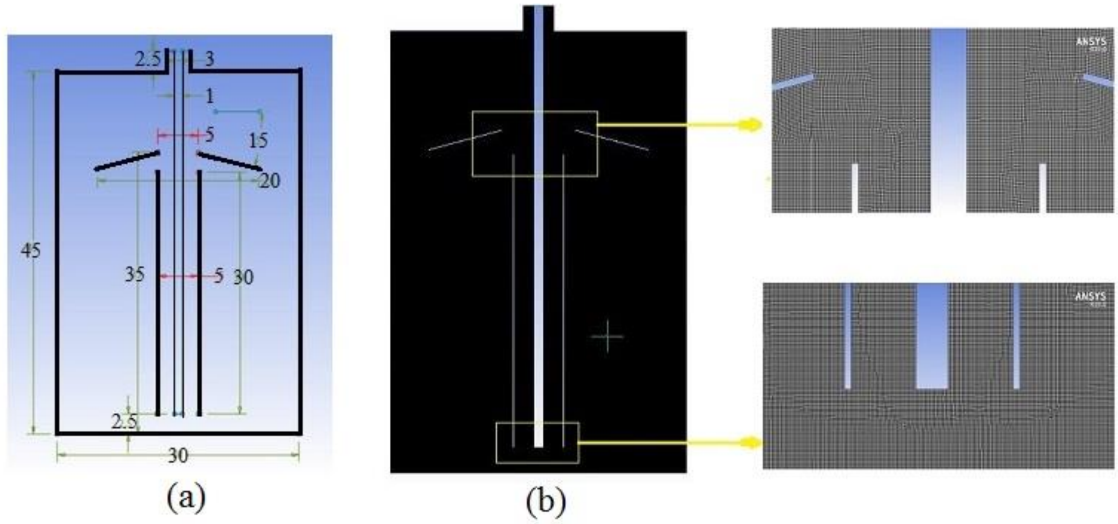

Fig. 2: (a): Digester geometry used to optimize mixing ; (b) mesh used for simulation

Evaluation indexes

Sludge velocity

The velocity counter and streamlines were utilized in steps 1 through 6 of the simulation methodology to determine the inlet-gas velocity, draft tube, and conical hanging baffle characteristics. The uniformity of counters and streamlines, as well as their contribution to uniformity within the digester were considered (refer to Baveli Bahmaei *et al*. (2022) for more details). Sludge velocity was used as one of the validity indices for investigating the mixing quality in simulated gas-lift anaerobic digester and for deciding on appropriate inlet-gas velocity. The velocity was investigated in term of quantity and compared with the sludge sedimentation velocity. Wherever the velocity was less than the sedimentation velocity, it indicated that sludge particles would sediment in the digester.

Sludge velocity gradient

sludge velocity gradient used as a validation index for quality of mixing. This parameter is defined as a custom field function in ANSYS Fluent main menu by the following formula as Wu (2014) used. local velocity gradient for the mixture in multiphase flow using the SST k-w model defined as Wu (2014):

$$G_L = \sqrt{\frac{\rho\omega\beta^{*}k}{\eta}} \tag{1}$$

Where ρ and η are the density and the non-Newtonian viscosity for the liquid phase, respectively, w and k are the specific dissipation rate and the turbulence kinetic energy for the mixture, respectively, β^{*}= 0.09, and G_L is the local velocity gradient called the velocity gradient hereafter.

Turbulence kinetic energy

Use turbulence kinetic energy as one of the indexes that investigate mixing quality in simulation results. Turbulence kinetic energy defined as:

$$G_k = -\rho\overline{u_i'u_j'}\frac{\partial u_j}{\partial x_i} \tag{2}$$

Reynolds stresses, $-\rho\overline{u_i'u_j'}$, by employs the Boussinesq hypothesis to relate the mean velocity gradient defined as:

$$-\rho\overline{u_i'u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\mu_t\frac{\partial u_i}{\partial x_i}\right)\delta_{ij} \tag{3}$$

Where μ_t is the turbulent viscosity, k is the turbulence kinetic energy, and u is the velocity that velocity component defined as:

$$u_i = \overline{u_i} + u_i' \tag{4}$$

Where $\overline{u_i}$ and u_i' are the mean and fluctuating velocity components respectively (i=1, 2, 3).

Sludge eddy viscosity

Uses sludge eddy viscosity as another index to investigate mixing quality in simulation results. Sludge eddy viscosity is the proportionality factor in describing the turbulent energy transfer in the form of moving eddies, giving rise to tangential stresses. Eddy viscosity defined as (menter, 1993):

$$v_T =: \frac{\mu_T}{\rho} = \frac{k}{\omega} \tag{5}$$

Mixing Energy Level

The mixing energy level (MEL) can be estimated Eq. 6 (stukenberg *et al*., 1992):

$$MEL = \frac{E}{V} \tag{6}$$

where V denotes the effective volume of the digester and E denotes the energy consumption.

Energy consumption for the gas-sparging was evaluated based on the power input calculation (McFarland, 2001):

$$E = P_1 \cdot Q \cdot ln\left(\frac{P_2}{P_1}\right) \tag{7}$$

where Q denotes the gas flow rate; and P1 and P2 are the absolute pressure in the tank headspace and at the gas-sparging inlet, respectively.

Particle image velocimetry

According to the methods of Raffel *et al*. (1998) and Dawkins *et al*. (2012) the particle image velocimetry (PIV) process involves taking two images (I1 and I2) separated by a time distance of Δt. Both images, subsequently, were divided into smaller regions, also known as sub-windows, interrogation-windows or interrogation-regions. Each sub-window in the first image is compared with the corresponding sub-window in the second image. The sub-window number i, j in the first image is denoted as $I^{i,j}_1$, while the corresponding sub-window in the second image is denoted as $I^{i,j}_2$.Hereafter, performed a search algorithm to identify a displacement of the pattern in $I^{i,j}_1$. To do this, the squared Euclidean distance between the two sub-windows has defined as:

$$R_e(s.t) = \sum_{m=0}^{M-1}\sum_{n=0}^{N-1}\left[I_1^{i.j}(m.n) - I_2^{i.j}(m-s.n-t)\right]^2 \qquad (8)$$

It means that, calculates the sum of the squared difference between the sub-windows for every possible overlap of the sub-windows. In other words, this means that looking algorithm for the position where the sub-windows were the “least unlike”. If expanded Eq. 8 the square parentheses on the right-hand side, it would result to:

$$\begin{aligned} R_e(s.t) &= \sum_{m=0}^{M-1}\sum_{n=0}^{N-1}\left[I_1^{i.j}(s.t) - I_2^{i.j}(m-s.n-t)\right]^2 \qquad (9)\\ &= \sum_{m=0}^{M-1}\sum_{n=0}^{N-1} I_1^{i.j}(m.n)^2 - 2I_1^{i.j}(s.t)\cdot I_2^{i.j}(m-s.n-t) \\ &\quad + I_2^{i.j}(m-s.n-t)^2 \end{aligned}$$

It should be noted that the first term, $I^{i,j}_1\ (m,n)^2$, is merely a constant since it does not depend on s and t. The last term, $I^{i,j}_2\ (m – s, n - t)^2$ is seen to depend on s and t, but that is just dependent on the second image. So, to sum up, only the middle term actually deals with both our images, and as a matter of fact this term (without the -2) is usually referred to as cross-correlation (or circular cross-correlation) and defined as:

$$R(s.t) = \sum_{m=0}^{M-1} \sum_{n=0}^{N-1} I_1^{i.j}(m.n) \cdot I_2^{i.j}(m - s.n - t) \quad (10)$$

Results and discussion

The mixing conditions in the digester with were investigated using different inlet-gas velocities. Simulations were performed using inlet-gas velocities of 0.05, 0.1, 0.2, 0.3, 0.4, 0.5 and 0.6 ms^{-1} to study the mixing in the cylindrical digester, the details of which are indicated in Fig. 2.

Investigate evaluation indexes

The values of the investigated indexes and Mixing Energy Level (MEL) for each of gas-inlet velocity are in Table 1.

Table 1: Evaluated indexes and MEL for verifying the mixing quality in gas-lift anaerobic digester by different inlet-gas velocity.

Inlet-gas velocity (ms^{-1})	Sludge velocity (ms^{-1})			Turbulence kinetic energy (m^{2}s^{-2})		Average velocity gradient (s^{-1})		Sludge eddy viscosity (Pa s)		MEL
	min. E-6	Ave.	max.	min. E-14	max.	min. E-6	max.	min. E-17	max.	
0.05	2.23	0.0236	0.30	1.0	3.8E-07	6.6	0.07	5.85	3.0E-08	0.505
0.1	10.72	0.0291	0.43	1.0	8.3E-07	18	0.14	5.91	8.0E-07	1.01
0.2	1.40	0.0287	0.66	1.0	64E-07	29	0.29	7.75	1.8E-05	2.02
		0.0								3.03
0.3	2.61	322	0.83	8.1	0.011	359	285.23	64.50	73E-05	
0.4	3.92	0.0375	1.16	120	0.17	1398	449.68	974.12	0.65	4.04
0.5	2.26	0.0443	1.29	4400	0.21	8370	536.97	34911.40	0.63	5.05
0.6	9.53	0.0453	1.49	1900	0.26	5461	672.24	14858.50	0.74	6.06

Sludge velocity

Table 1 shows the minimum, average, and maximum values of sludge velocity for different inlet-gas velocities. The minimum sludge velocities were achieved in local and face options. The maximum velocity appears inside the draft tube, while the minimum value appears near the digester walls and bottom. The maximum velocity varies from 0.3 to 1.49 ms^{-1} for inlet velocities from 0.05 to 0.6 ms^{-1}, while the average velocity varies only by about 0.022 ms^{-1}. This indicates that the velocity of particles in all internal parts of the digester does not increase proportionally with the increase in the inlet-gas velocity. This could be due to the formation of short-circuiting in the digester in areas where more mixing takes place, as sludge is a non-Newtonian fluid and more mixing causes its viscosity to decrease further.

Since the maximum sludge particles sedimentation velocity is 47E-6 ms^{-1} (Baveli Bahmaei *et al.*, 2022), to prevent particle sedimentation, the minimum sludge velocity should be greater than 47E-6 ms^{-1}. However, this goal is not achieved thoroughly at any of the inlet-gas velocities when considering the minimum fluid velocities at different inlet-gas velocities. On the other hand, increasing the inlet-gas velocity in gas-lift anaerobic digesters is limited due to the biological nature of anaerobic digestion. Therefore, a balance must be struck between increasing the mixing rate and reducing the particle sedimentation to maintain the conditions that prevent disruption of the biological process of anaerobic digestion.

Turbulence kinetic energy

The minimum and maximum values of turbulence kinetic energy for different inlet-gas velocities showed in Table 1. Minimum of turbulence kinetic energy varies between 1E-14 and 44E-12 m^2s^{-2} in inlet-gas velocity of 0.05 and 0.5 ms^{-1} respectively and maximum of it varies from 3.8E-7 until 0.26 m^2s^{-2} in 0.05 until 0.6 ms^{-1}. Turbulence kinetic energy produced by the first three inlet-gas velocities (0.05, 0.1, and 0.2 ms^{-1}) is very low, and for the last three of them (0.4, 0.5, and 0.6 ms^{-1}) is high and closed together While for inlet-gas velocity of 0.3 ms^{-1} it has an intermediate value. Result of turbulence kinetic energy for inlet-gas velocity of 0.3 ms^{-1} presented in Fig. 3. High turbulence kinetic energy caused more intense mixing and destruction of flocs, which disrupt the anaerobic digestion process.

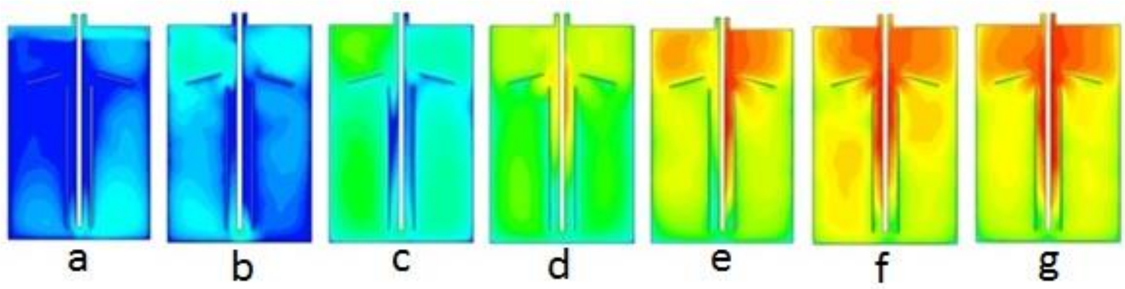

Fig. 3: Turbulence kinetic energy contours (logarithmic color) for different inlet-gas velocities; (a): 0.05, (b): 0.1, (c): 0.2, (d): 0.3, (e): 0.4, (f): 0.5, (g): 0.6 ms^{-1}

Average velocity gradient

The minimum and maximum values of the average velocity gradient for different inlet-gas velocity showed in Table 1. The minimum average velocity gradient varies from 6.6E-12 to 84E-10 s^{-1} for inlet-gas velocities of 0.05 to 0.5 ms^{-1}. The maximum average velocity gradient varies from 0.07 to 672.24 s^{-1} in the inlet-gas velocity of 0.05 to 0.6 ms^{-1}. The average velocity gradient produced by the first three inlet-gas velocities (0.05, 0.1, and 0.2 ms^{-1}) is lower and for the last three of them (0.4, 0.5 and 0.6 ms^{-1}) is high and closed together, while for inlet-gas velocity of 0.3 ms^{-1}, it has an intermediate value. Result of the average velocity gradient for inlet-gas velocity of 0.3 ms^{-1} presented in Fig. 4. Turbulence kinetic energy generation caused by the average velocity gradient, and the velocity gradient results were similar to it.

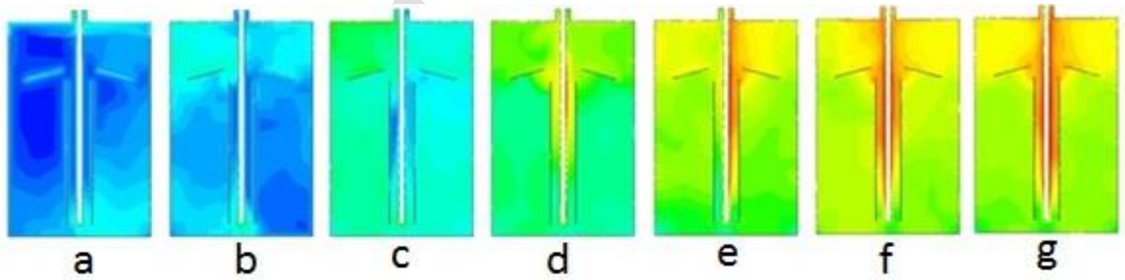

Fig. 4: Average velocity gradient contours (logarithmic color) for different inlet-gas velocities; (a): 0.05, (b): 0.1, (c): 0.2, (d): 0.3, (e): 0.4, (f): 0.5, (g): 0.6 ms^{-1}

Sludge eddy viscosity

Sludge eddy viscosity is the proportionality factor describing the turbulent energy transfer as a moving eddies result, giving rise to tangential stresses. The minimum and maximum values of sludge eddy viscosity for different inlet-gas velocities showed in Table 1. The minimum of sludge eddy viscosity varies from 5.85E-17 to 14.86E-14 Pa s in inlet-gas velocity of 0.05 to 0.6 ms^{-1}, and a maximum of it varies from 3.0E-8 to 0.74 Pa s in 0.05 to 0.6 ms^{-1}. Sludge eddy viscosity produced by the first four inlet-gas velocities (0.05, 0.1, 0.2, and 0.3 ms^{-1}) is lower, and for the last three of them (0.4, 0.5 and 0.6) is high and closed together**.** Higher eddy viscosity indicates a

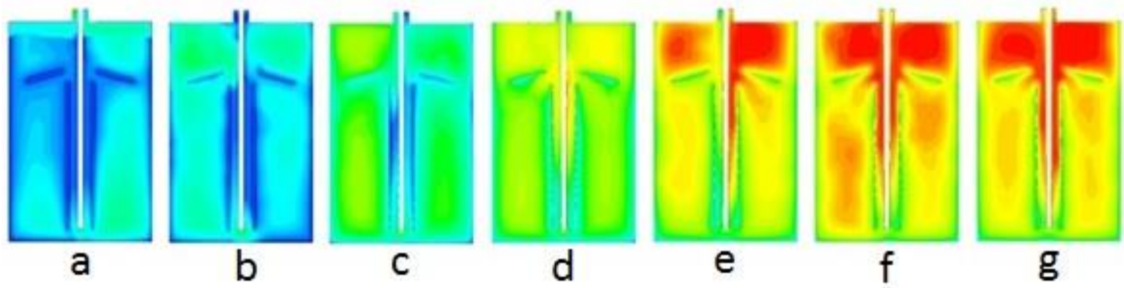

Fig. 5: Sludge eddy viscosity contours (logarithmic color) for different inlet-gas velocities; (a): 0.05, (b): 0.1, (c): 0.2, (d): 0.3, (e): 0.4, (f): 0.5, (g): 0.6 ms^{-1}

high amount of moving eddies and high tangential stresses in the sludge that can lead to the destruction of flocs and disrupt digestion as a biological process. Therefore, in terms of sludge eddy viscosity index, an inlet-gas velocity of 0.3 ms^{-1} was appropriate. Result of sludge eddy viscosity for inlet-gas velocity of 0.3 ms^{-1} presented in Fig. 5.

Select appropriate inlet-gas velocity

The investigation of the evaluation indices revealed that there must be a balance between mixing intensity and sludge sedimentation. Higher mixing intensity can result in broken floc and impaired anaerobic digestion. If a high inlet-gas velocity is selected for mixing, it can disrupt the biological process of anaerobic digestion. On the other hand, if the velocity is too low, the particle sedimentation rate will increase, and proper mixing will not occur.

The investigation of sludge velocity, turbulence kinetic energy, average velocity gradient, and eddy viscosity showed that an inlet-gas velocity of 0.3 ms^{-1} is more appropriate. The results of CFD simulations for the investigated evaluation indices at an inlet-gas velocity of 0.3 ms^{-1} are showed in Fig. 6.

The sludge velocity counter presented in Fig. 6 indicates that in most of the digester zone (zones 4 and 5 with yellow and red colors), the particle velocity is greater than 1.75E-3 ms^{-1}. Considering the maximum sludge sedimentation velocity for the largest sludge particle (47 E-6 ms^{-1} for particles size of 2 mm) (Baveli Bahmaei *et al*., 2022), particle sedimentation in the digester is very low. Even in zone 3 with green color, the sludge velocity was larger than 9.9E-5 ms^{-1}. Only in zones 2 and 1 where sludge velocity is lower than 9.9E-5 ms^{-1}, there is a possibility of sedimentation of particles larger than 0.85 mm, which comprise 17% of the total particles in the sludge (Baveli Bahmaei *et al.*, 2022). However, zones 1 and 2 cover a very small percentage of the digester volume, indicating good mixing conditions.

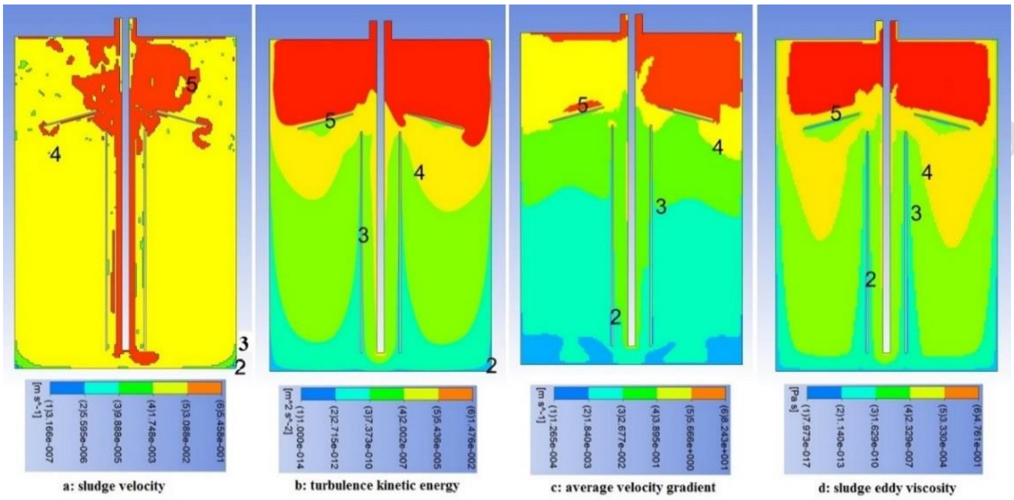

Fig. 6: Results of CFD simulations for investigated evaluation indexes by inlet gas velocity of 0.3 ms^{-1}.

Gas-sparging intensity determines the amount of injected biogas for mixing and is consequently an important operational parameter to assess. Based on the compressor's capacity, the injected biogas flow rate for inlet-gas velocity of 0.3 ms^{-1} was calculated to be 0.085 m^3h^{-1} in the studied digester. The realistic 0.085 m^3h^{-1} scenario yielded a *MEL* of 3.3 Wm^{-3}, which was close to 2.2 Wm^{-3} reported in another full-scale gas-mixed digester (Dapelo and Bridgeman, 2018) but was still much lower than the recommended range of 5–8 Wm^{-3} for proper mixing (U. EPA, 1979). To match the recommended range, the inlet-gas velocity should be increased to over 0.7 ms^{-1}. Such change requires more investment, more technical adjustment and much larger energy consumption and may challenge the biogas production process in the studied digester. Therefore, intensifying the inlet-gas velocity seems not an efficient strategy to enhance flow and mixing, and the recommended *MEL* criterion appears unsuitable for the studied digester.

Particle image velocimetry results

To verify CFD simulation results a transparent digester was constructed with the ability to take photos from the outside. The transparent digester is shown in Fig. 7a. Polymethyl methacrylate with a thickness of 1.5 mm was used to build a pilot-scale digester with characteristics obtained from the CFD simulation results.

(a) (b)

Fig. 7: built, and loaded digester: (a): built transparent digester by Polymethyl methacrylate, (b) loaded digester by municipal wastewater sludge.

After selecting the inlet-gas velocity of 0.3 ms^{-1} as the most appropriate inlet-gas velocity, the particle image velocimetry (PIV) was performed. Due to the very dark color of the sludge (see Fig. 7b) and the indistinct particles in the images, the glitter particles were used for PIV.A narrow strip along the height of the digester was considering for PIV. The calculated sludge velocity, average velocity gradient, and sludge streamlines are showed in Fig.8. The average velocity gradient (Fig. 8a) varies from 1.8E-6 to 34.3E-6 s^{-1}, while sludge velocity (Fig .8b) varies from 0 to $1.1*10^{-3}$ ms^{-1}. The maximum value of average velocity gradient and sludge velocity occurred between 20 to 35 cm distance from the top of the digester, and the streamline distance in this zone is maximum. As shown in Fig. 8b, the sludge velocity in most parts of the digestion wall length is greater than the minimum sludge velocity achieved from the simulations, indicating that particles sedimentation does not occur**.** Observation of the velocity counter obtained from the PIV shows that the lowest velocity is at the junction of the wall to the bottom of the digester (Fig. 8b), and the streamlines (Fig. 8c) show that there are no streamlines in this area, confirming the results of CFD simulations.

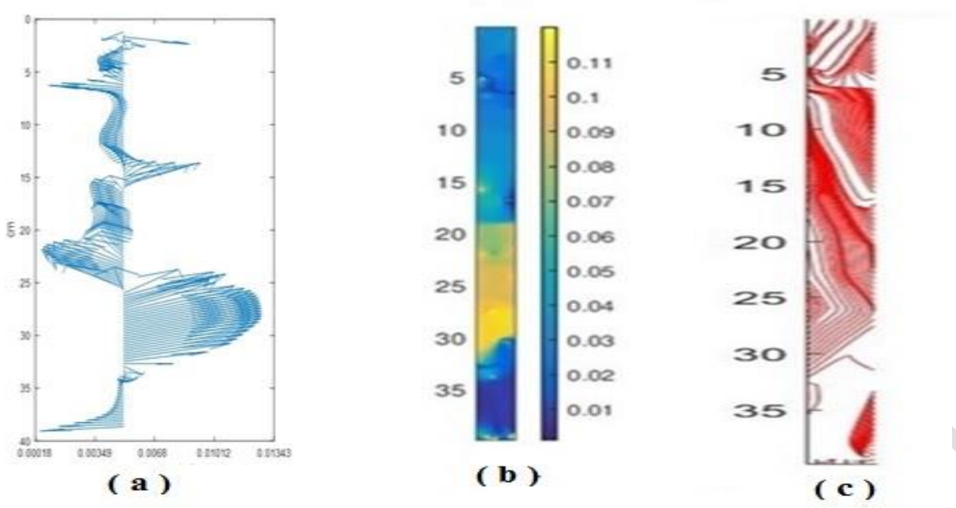

Fig. 8: Results of particle image velocimetry (PIV) for inlet gas velocity of 0.3 ms^{-1}: (a): average velocity gradient (s^{-1}), (b): sludge velocity (cms^{-1}), (c): streamline of particles in sludge.

Conclusion

The present study aimed to optimize mixing in gas-lift anaerobic digesters of municipal sewage sludge through the implementation of a model that simulated, optimized, and confirmed the simulation process. Computational Fluid Dynamics (CFD) by ANSYS Fluent software was utilized to perform simulations of the digester, which comprised a cylindrical tank with flat bottom and height to diameter ratio equal 1.5 (45 to 30 cm), draft tube and conical hanging baffle.

To optimize mixing in the digester, different inlet-gas velocities were investigated, and sludge particle velocity, the gradient of sludge particle velocity, the turbulence kinetic energy, and the eddy viscosity of sludge particles then evaluated as indexes. The evaluation indexes contours were analyzed to determine the appropriate velocity for optimal mixing, which was found to be 0.3 ms^{-1}.

Based on the simulated digestion characteristics, the selected inlet-gas velocity, and particle sedimentation velocity in sludge, it was expected that the sedimentation of the particles in the digester would not occur except for large sludge particles in a small triangular section near the junction of the wall to the bottom of the digester. Subsequently, a transparent anaerobic digester was constructed and loaded with municipal sewage sludge, operating at the optimal inlet-gas velocity of 0.3 ms^{-1}. To validate the CFD simulation outcomes, Particle Image Velocimetry (PIV) was employed to calculate sludge velocity, average sludge gradient, and streamlines. According to the results of the particle image velocimetry (PIV),the sludge velocity in most parts of the digestion wall length is greater than the sludge velocity minimum achieved from the simulations, and the velocity counter obtained from the PIV shows that the lowest velocity is at the junction of the wall to the bottom of the digester and streamlines also showed that there are no streamlines in this area. The PIV method was used to verify the CFD simulation results, which showed sufficient agreement between both methods. The results showed that the model used for simulating, optimizing, and verifying the simulation process was successful. It can be recommended for similar gas-lift anaerobic digesters, which consist of a cylindrical tank with a flat bottom and a height-to-diameter ratio of 1.5. The draft tube diameter should be 0.2 times the digester diameter, while the draft tube height should be 0.75 times the fluid height. The conical hanging baffle distance from the fluid level should be equal to 0.125 times the fluid height, and its outer diameter should be 2/3 of the digester diameter.

References

ANSYS-Fluent Inc. 2008. Fluent 12.0. ANSYS-Fluent Inc, Lebanon, N.H.

Baveli Bahmaei, D., Ajabshirchi, Y., Abdollah poor, Sh., Abdanan Mehdizadeh, S. 2022. Step-by-step Simulation of Gas-lift Anaerobic Digester of Municipal Wastewater Sludge. Iranian Journal of Biosystem Engineering 53 (1): 91- 108. (In Persian).

Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Coleman, H. & Raad, P. E. 2008. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. Journal of Fluids Engineering 130. 078001.

Dapelo, D., Alberini, F., Bridgeman, J. 2015. Euler-Lagrange CFD modelling of unconfined gas mixing in anaerobic digestion. Journal of Water Researches 85: 497–511.

Dapelo, D., and Bridgeman, J. 2018. Euler-Lagrange Computational Fluid Dynamics simulation of a full-scale unconfined anaerobic digester for wastewater sludge treatment. Advances in Engineering Software 117: 153–169.

Dawkins, M. S., Cain, R., Roberts, S. J. 2012. Optical flow, flock behavior and chicken welfare. Animal Behavior 84 (1): 219-223.

Hu, Y., Zhang, Sh., Wang, X., Peng, X., Hu, F., Wang, Ch., Wu, J., Poncin, S., Li, H. Z. 2021. Visualization of mass transfer in mixing processes in high solid anaerobic digestion using Laser Induced Fluorescence (LIF) technique. Waste Management 127: 121–129.

Karim, Kh., Varma, R., Vesvikar, M. & Al-Dahhan, M. H. 2004. Flow pattern visualization of a simulated digester. Journal of Water Researches 38: 3659–3670.

Karim, Kh., Klasson, k. T., Hoffmann, R., Drescher, S. R., DePaoli, D. W. & Al-Dahhan, M. H. 2005. Anaerobic digestion of animal waste: Effect of mixing. Journal of Bioresource Technology 96: 1607–1612.

Karim, K., Thoma, G. J. & Al-Dahhan, M. H. 2007. Gas-lift digester configuration effects on mixing effectiveness. Water Researches 4: 3051-3060.

McFarland, M.J. 2001. Biosolids Engineering. McGraw-Hill Education, New York.

Menter, F. R. 1993. Zonal Two Equation k-cl, Turbulence Models for Aerodynamic Flows. 24th Fluid Dynamics Conference. Orlando, Florida.

Meynell, P. J. 1976. Methane: Planning a Digester. London: Prism Press: 55–57.

Raffel, M., Willert, C. E., Kompenhans, J. 1998. Paeticle Image Velocimetry, A Practical Guide. Springer Verlage, first edition.

Sawyer, C. N. and Grumbling, A. M. 1960. Fundamental consideration in high-rate digestion. Inc. Sewage Engineering Division. ASCE: 86–92.

Stukenberg, J. R., Clark, J. H., Sandino, J., Naydo, W. R., 1992. Egg-shaped digesters: from Germany to the U.S. Water Environment Technology 4: 42–51.

U. EPA. 1979. Process design manual for sludge treatment and disposal, Center for environmental research in formation technology transfer. pp.

Varma, R., Al-Dahhan, M. H. 2007. Effect of sparger design on hydrodynamics of a gas recirculation anaerobic bioreactor. Biotechnology Bioengineering 98 (6): 1146-1160.

Vesvikar, M. S. & Al-Dahhan, M. 2016. Hydrodynamics investigation of laboratory-scale Internal Gas-lift loop anaerobic digester using non-invasive CAPRT technique. Biomass and Bioenergy 84: 98-106.

Wei, P., Uijttewaal, W., Spanjers, H., Lier, J. B., Kreuk, M. 2023. Optimising flow and mixing in a full-scale gas-mixed anaerobic digester by integrating sludge rheological data using computational fluid dynamics. Chemical Engineering Journal 468. https://doi.org/10.1016/j.cej.2023.143647

Wu, B. 2010. CFD simulation of gas and non-Newtonian fluid two-phase flow in anaerobic digesters. Journal of Water Researches 44: 3861-3874.

Wu, B. 2014. CFD simulation of gas mixing in anaerobic digesters. Journal of Computers and Electronics in Agriculture 109: 278–286.
Yang, J., Yang, Y., Ji, X., Chen, Y., Guo, J., & Fang, F. 2015. Three-Dimensional Modeling of Hydrodynamics and Biokinetics in EGSB Reactor. Journal of Chemistry.

بهینه‌سازی هم‌زنی در هاضم بی هوازی گاز- بالابر لجن فاضلاب شهری

داود باولی بهمئی[۱]، یحیی عجب شیرچی[۲]، شمس اله عبداله پور[۳*]، **سامان آبدانان مهدی زاده[۴]**

۱. دانشجوی دکتری، گروه مهندسی بیوسیستم، دانشکده کشاورزی، دانشگاه تبریز، تبریز، ایران
۲. استاد، گروه مهندسی بیوسیستم، دانشکده کشاورزی، دانشگاه تبریز، تبریز، ایران
۳. دانشیار، گروه مهندسی بیوسیستم، دانشکده کشاورزی، دانشگاه تبریز، تبریز، ایران
۴. استادیار، گروه مهندسی مکانیک بیوسیستم، دانشکده مهندسی زراعی و عمران روستائی، دانشگاه علوم کشاورزی و منابع طبیعی خوزستان، اهواز، ایران
(*- نویسنده مسئول: shams@tabrizu.ac.ir)

چکیده

این تحقیق با هدف بهینه‌سازی هم‌زنی در هاضم‌های بی‌هوازی گاز-بالابر لجن فاضلاب شهری انجام شد، زیرا هم‌زنی یکنواخت برای ارتباط مؤثر بین باکتری‌های متانوژن و مواد مغذی مهم است. نمونه‌برداری لجن فاضلاب شهری در تصفیه‌خانه غرب اهواز (چنیبه) در تابستان ۱۴۰۱ انجام شد. یک مدل برای شبیه‌سازی، بهینه‌سازی و تایید فرآیند شبیه‌سازی با استفاده از دینامیک سیالات محاسباتی (CFD) توسط نرم افزار ANSYS Fluent 19.0 ارائه شد. سرعت گاز ورودی به هاضم تعیین شد و یک لوله گاز- بالابر و بافل آویزان مخروطی به طرح هاضم اضافه شد. سرعت‌های مختلف گاز ورودی برای بهینه‌سازی اختلاط در هاضم مورد بررسی قرار گرفت و شاخص‌های ارزیابی مانند سرعت ذرات لجن، گرادیان سرعت ذرات لجن، انرژی جنبشی تلاطم و ویسکوزیته گردابی ذرات لجن مورد ارزیابی قرار گرفت. سرعت بهینه گاز ورودی ۰٫۳ ms^{-1} تعیین شد. نتایج شبیه‌سازی با استفاده از روش سرعت‌سنجی تصویری ذرات (PIV) تأیید شد و درصد همبستگی کافی بین کانتورهای CFD و PIV وجود داشت (۹۸٫۸٪ در محل اتصال دیواره به کف هاضم). نتایج نشان داد که مدل مورد استفاده برای شبیه‌سازی، بهینه‌سازی و تأیید فرآیند شبیه‌سازی موفق بوده است و می‌توان آن را برای هاضم‌های بی‌هوازی گاز- بالابر استوانه‌ای شکل با نسبت ارتفاع به قطر ۱٫۵، نسبت قطر لوله گاز- بالابر به قطر هاضم ۰٫۲، نسبت ارتفاع لوله گاز- بالابر به ارتفاع سیال ۰٫۷۵، فاصله بافل آویزان مخروطی از سطح سیال ۰٫۱۲۵ برابر ارتفاع سیال و قطر بیرونی بافل به قطر هاضم ۲/۳ توصیه کرد.

واژگان کلیدی: دینامیک سیالات محاسباتی (CFD)، هضم، سرعت سنجی تصویر ذرات (PIV)، شبیه‌سازی.