CFD Simulation of Two-Phase Flows in Anaerobic Digester

Jianyuan Chen¹, Anpeng Chen¹, Jiunnhaur Shaw¹, Chewei Yeh¹, *, and Shingder Chen²

¹Center for Measurement Standards/Industrial Technology Research Institute, Taiwan
²Material and Chemical Research Laboratories/Industrial Technology Research Institute, Taiwan

*Corresponding author e-mail: jerwei2003@itri.org.tw

Abstract. In recent years, the study of Anaerobic Digestion (AD) has become one of the important technologies in the development of biogas production as renewable energy source. It can reduce the emissions of greenhouse gases and improve management of solid wastes. The production of biogas is influenced by many parameters including temperature, mixing, pH and hydraulic retention time etc. In this study, the mathematical model and numerical simulation of the turbulent, two-phase flow of liquid and gas in a Anaerobic Digester were conducted by Computational Fluid Dynamics. The hydrodynamic behaviours of the gas-lift and mechanical mixing digesters were investigated, such as velocity field, dynamic viscosity and optimal mixing type.

1. Introduction

Anaerobic Digestion (AD) is a spontaneous process mediated by micro-organisms without requiring advanced dehydrate procedure or further chemical extraction process. Micro-organisms convert biomass into biogas which is a mixture of mainly CO₂ and CH₄. AD is one of the methods that often used in waste treatment and the bi-product biogas has been considered as useful renewable energy. Mixing technique is a useful way to improve the effect of AD. During the mixing process, the scum formation and gradients of pH and temperature should be avoided. Besides, over mixing might damage the microbe in the digester and cause extra energy usage. The mixing parameters such as mixing time, temperature, PH and mixing agitation would play important roles in the AD process. The mixing methods include jet-stream mixing, mechanical mixing and gas-lifting mixing, etc. Mechanical mixing design is often accomplished by helical ribbon and anchor impellers to mix the solid and liquid. The design and operation efficiency might be influenced by electricity usage and construction cost during the process. Gas-lifting is used to pump the gas into digester to mix the solid and liquid.

The mixtures in AD digester are non-Newtonian fluids. The techniques of particle characterization including DSCT (Dual Source CT) and RPT (Radioactive Particle Tracking) were used to investigate the digester design factor and particle mixing property in AD digester [1-3]. The crystal clear digester was built up and observed by Particle Image Velocity (PIV) to measure the velocity and flow in the digester [4]. In recent years, the Computational Fluid Dynamics (CFD) technique is used to compute fluid flow and design the geometry of the blade agitation in the reaction tank [5-6]. The results show that CFD simulation can be used to predict the radial and axial velocity correctly in the digester. The simulated and experimental flow field results are consistent [7]. In this study, CFD is used to simulate
the distribution of velocity field, uniformity, and dynamic viscosity and evaluate the gas lift and mechanical mixing methods in the anaerobic digester.

2. Numerical Simulation

2.1. Basic model setting
In this study, the CFD software ANSYS Fluent Ver. 16.0 based on the finite-volume methods is used to analyze the fluid flow in the digester. The calculated zone is divided into many staggered grids. The discrete Reynolds-averaged Navier-Stokes equations in each grids are solved by iterative processes and solve for velocity, pressure, and temperature values. In order to fully simulate the turbulent flow in the digester, the $k$-$\varepsilon$ RNG turbulent model is conducted in this simulation. $k$-$\varepsilon$ RNG turbulent model has good accuracy in simulating complicated flow such as swirling flow and separation flow, so we use this model to solve for quasi-steady flow field in this study. On the purpose of simulating rotating machinery numerically, ANSYS provides four rotating models: Multiple Reference Frame Model (MRF), Mixing Plane Model (MPM), Sliding Mesh Model (SMM), and Dynamic Mesh Model (DMM). We choose Sliding Mesh Model (SMM) to solve for transient mixing flow solution in constant rotational speed.

The VOF model is one of the multiphase flow simulation methods especially for free surface flow of liquid and gas inside the digester. At first, $f$ as the integral of a fluid’s characteristic function in the control volume is defined. When the cell is full, $f=1$; and when there is a fluid interface in the cell, $0 < f < 1$. $f$ is a discontinuous function, its value jumps from 0 to 1 when the argument moves into interior of traced phase. Governing equation is shown below [6].

$$\frac{\partial f}{\partial t} + \nabla \cdot (vf) = 0$$

We assume that the total solid is over 5% in the digester and characterize the Non-Newtonian fluid as the pseudoplastic, isothermal and incompressible fluid which has the negative correlation with shear rate and well-fluidity. The viscosity is described by Power Laws Model as shown below [8]. We set consistency index $k = 2.4$ (Pa-s$^n$), power-law index $n$ is 0.38, the minimum viscosity is 0.34 (Pa-s), and the maximum viscosity is 2.46 (Pa-s). The fluid density is 999.43 (kg/m$^3$).

$$\tau = \mu_k \gamma^n$$

2.2. Numerical model
The geometry and mesh distribution of mechanical mixing digester is shown in Figure 1. The outer wall surface is set as a static wall surface and the gas-liquid free liquid surface is arranged in the top of the digester tank. The spiral stirring shaft is placed on the central axis and enclosed by the near fluid to become a rotation zone. The height and the diameter of the digester are 9 m and 3.8 m. The height and the diameter of the impeller are 5.86 m and 1.9 m. The unstructured mesh is constructed by the Fluent Mesh software. The total grids number is 516,461. The gas pressure was set 1 atmosphere. The wall roughness is ignored. The geometry and mesh distribution of Gas-Lifting digester is shown in Figure 2. The height and the diameter of the digester are 6.8 m and 3.4 m. The height of the upper air zone is 0.5 m. The diameter and the height of the center shaft are 0.5 m and 4.5 m. The total mesh number is 416,461.

Figure 1. Geometry and Mesh Distribution in Mechanical Mixing Digester
3. Results and Discussion

3.1. Mechanical Mixing Digester

Figure 3 and Figure 4 demonstrated the flow velocity distribution changed with time and streamline in the reaction tank at rotational speed 10 rpm. The radial and axial flows occurred simultaneously with the rotational speed. The flow field reached quasi-steady in 17.85 s. The flow generated by the rotation of the auger blade was recirculated from the bottom to the outside region. The higher velocity fluid zone was near the stirring rod, especially at the edge of the blade. The fluid velocity was low at the bottom of the reactor and the upper corner region. These zones are almost stagnant and can be called dead zones that leads difficulty in fully mixing.

The shear stress determined the power consumption of the agitation. For the pseudoplastic fluid, the viscosity of the fluid decreased as the shear rate increases. Figure 5 showed the shear rate distribution in the vertical plane x=3 m. The shear rate was mainly distributed in the flow region centered on the auger under the driving of the spiral blade. And the stress rate of the fluid outside the blade region decreased rapidly as the radius changed. The radius of the spiral blade was small resulting in a small flow rate below the reaction tank. The shear rate was lower in the bottom of the reactor and the vicinity of the shaft center. It meant that the energy transfer is the slowest in this area.
Figure 5. Shear rate distribution changes with radius in the vertical section x=3 m

Figure 6 showed the distribution of the stagnation zone at the rotational speed at 10 rpm and 20 rpm. It can be seen that as the spiral blade speed increased, the fluid at the bottom of the reactor did not change much. A more pronounced and faster mixing zone appeared in the vicinity of the stirrer blade and the fluid velocity at the outside was still slower. Therefore, it’s necessary to design improvements to the reactor structure and its auger blades to eliminate stagnant or slow flow areas and ensure that the fluids are in a fully mixed flow state.

Figure 6. The stagnation zone of fluid at different speeds in the reaction tank (speed below 0.05 m/s)

3.2. Gas-Lifting Digester

Figure 7 showed the velocity distribution of the central axis in the digester with different flow rates. When the gas was discharged from the bottom in the digester, the bubble density was small and the bubbles floated upward fast. When the bubbles reached the liquid surface, the gas moved outward and the liquid surface was limited by the boundary condition in the top area and moved down to the bottom of the digester. Eventually, two symmetrical vortices occurred on the purpose for well-mixing. Vortex center was located between the liquid surface and the upper central region of the digester. The increasing inlet gas flow caused more complicated vortex structures and high velocity zones.

Figure 7. Flow field in digester at different flow rates
Figure 8 showed that the dead zone (grey area) at different flow rates (7.3 m$^3$/h and 18.3 m$^3$/h) inside the digester. The flow velocity in the dead zone was lower than 0.05 m/s. The grey area occupied 44% of the total volume. Figure 9 demonstrated the viscosity at different flow rates inside the digester. Dynamic viscosity distribution was similar. Non-Newtonian fluid was not well-mixed easily because the lower velocity contributed to lower shear stress and higher dynamic viscosity. The bottom area in the digester was high viscosity area, the gas flow increasing and the liquid phase dynamic viscosity area is decreasing with smaller peak value, as shown in Figure 10 and Figure 11.

![Image](image1.png)

**Figure 8.** Dead zone in the reaction tank at different flow rates (grey area)

![Image](image2.png)

**Figure 9.** The dynamic viscosity distribution in the reaction tank

![Image](image3.png)

**Figure 10.** Dissipation process of different flow gases in the reaction tank with time (7.3 m$^3$/min)

![Image](image4.png)

**Figure 11.** Dissipation process of in the reaction tank with time (18.3 m$^3$/min)
4. Conclusion
This study is mainly focused on evaluation of the fluid behaviours and mixing efficiency during mixing process in digester. The results indicate two points as follow. (1) Observation of the velocity distribution in the digester. Higher velocity region is mainly concentrated near the spiral blade. The lower velocity region appears near the bottom of the digester and the liquid surface. In these areas, the transfer energy is the slowest that is not conducive to mixing and causes dead zone formation. (2) The flow field patterns are steady even with the mixed gas flow and the rotational speed of the blade in the digester. At this stage, the flow field characteristics in the digester are numerically simulated and the structure in the digester can be optimized to avoid these disadvantages for evenly mixing the fluid in the digester in the future.

References
[1] Karim, K., Varma, R., Vesvikar, M., Al-Dahhan, M.H., 2004. Flow pattern visualization of a simulated digester. Water Res. 38(17), 3659–3670.
[2] Karim, K., Hoffmann, R., Klasson, K.T., Al-Dahhan, M.H.,2005. Anaerobic digestion of animal waste: waste strength versus impact of mixing. Bioresour. Technol. 96 (16),1771–1781.
[3] Karim, K., Hoffmann, R., Klasson, K.T., Al-Dahhan, M.H.,2005. Bio-Energy Production from Anaerobic Digestion of Animal and Farm Wastes.
[4] James R W, Charles E W, Robert E W, Gregory L R. Fundamentals of momentum, heat, and mass transfer. John Wiley & Sons, Inc. 2008, 5th.
[5] Michael Harasek, Kittas C. Determination of mixing quality in biogas plant digesters using tracer tests and computational fluid dynamics, 2013,5,1269-1277.
[6] Michael Meister et al., Mixing non-Newtonian flows in anaerobic digesters by impellers and pumped recirculation, Advances in Engineering Software, 2018,115,194-203.
[7] Khursheed Karim et al., Gas-lift digester configuration effects on mixing effectiveness, Water Research, 2007, 3051-3060.
[8] Michael Meister, Massoud Rezavand, Christian Ebnec, Mixing non-Newtonian flows in anaerobic digesters by impellers and pumped recirculation, Advances in Engineering Software, 2018, 194–203.