Flow pattern visualization in a mimic anaerobic digester: experimental and computational studies

M.S. Vesvikar, R. Varma, K. Karim and M. Al-Dahhan

Bioprocessing and Bioengineering Laboratory, Chemical Reaction Engineering Laboratory, Department of Chemical Engineering, Washington University, St Louis, MO 63130, USA (E-mail: muthanna@che.wustl.edu)

Abstract Advanced non-invasive experiments like computer automated radioactive particle tracking and computed tomography along with computational fluid dynamics (CFD) simulations were performed in mimic anaerobic digesters to visualize their flow pattern and obtain hydrodynamic parameters. The mixing in the digester was provided by sparging gas at three different flow rates. The simulation results in terms of overall flow pattern, location of circulation cells and stagnant regions, trends of liquid velocity profiles, and volume of dead zones agree reasonably well with the experimental data. CFD simulations were also performed on different digester configurations. The effects of changing draft tube size, clearance, and shape of the tank bottoms were calculated to evaluate the effect of digester design on its flow pattern. Changing the draft tube clearance and height had no influence on the flow pattern or dead regions volume. However increasing the draft tube diameter or incorporating a conical bottom design helped in reducing the volume of the dead zones as compared to a flat bottom digester. The simulations showed that the gas flow rate sparged by a single point (0.5 cm diameter) sparger does not have appreciable effect on the flow pattern of the digesters.

Keywords Anaerobic digester; computer automated radioactive particle tracking; computer tomography; computational fluid dynamics; flow pattern

Introduction

Biomass has a great potential to be used for energy generation. Methane generated from biomass is a renewable energy source. Hence, there is a great emphasis to improve the technology and methods for obtaining methane from biomass. Different types of biodigesters are used for methane generation. However, digester failure rates are quite high. In many of these cases failure was due to deposition and stratification, which can be attributed to the inefficient and insufficient mixing provided. Mixing is essential in digesters for their optimal performance and operation. In spite of its important role in the digester performance, the information available in the literature is confusing and inconclusive. Many of the findings are contradictory, viz., Sawyer and Grumbling (1960) and Meynell (1976) concluded mixing is required in anaerobic digesters. Ben-Hassan et al. (1985) reported mixing is not required in these systems whereas according to Dague et al., 1970 intermittent mixing is best for digesters. In view of these points it becomes necessary to investigate the relation between mixing and digester performance so that the performance of the digester can be improved and predicted with sufficient accuracy, which will also help in the design of digesters in an efficient and economical way. Thus the objective of this work was to study the hydrodynamics of a mimic digester mixed with gas recirculation.

Common experimental techniques used for visualizing flow pattern cannot be used for digesters due to their opaque nature. Thus advanced non-invasive techniques like computer automated radioactive particle tracking (CARPT) for obtaining flow pattern and velocity data and computed tomography (CT) for obtaining phase hold up data (Karim et al., 2004) must be used to quantify the hydrodynamic parameters in such systems. In absence of these tools, computational fluid dynamics (CFD) proves to be a
valuable and efficient tool to understand and evaluate the fluid dynamics of multiphase flow systems. Applying CFD can simulate the flow pattern in a selected digester. From these experiments and simulations, hydrodynamic parameters such as velocity, turbulence, hold up profiles, volume of dead zones etc. can be quantified and used in the design selection and optimization of anaerobic digesters (Mehul and Al-Dahhan, 2005).

The summary of experimental (Karim et al., 2004) and computational work (Mehul and Al-Dahhan, 2005) presented in this paper evaluates three-dimensional CFD simulations and compares the results to experimental data obtained by CARPT. The effects of various design and operating parameters on the digester flow pattern are also studied.

**Digester geometry**

Karim et al. (2004) applied CARPT and CT on a mimic flat bottom digester mixed by gas recirculation shown in Figure 1(a). The slurry used in the Karim et al. experiments was a 50-50 mixture of primary and secondary municipal sludge contained 50 gm/l solids. Sodium azide was added to the slurry to stop the activity of microorganisms. Air was sparged with a pipe sparger located at the center of the tank; the single opening of sparger was 0.5 cm in diameter. Air was sparged at three different gas flow rates (volumetric air flow rate of 28.32 l/h, 56.64 l/h and 84.96 l/h) with superficial gas velocity (based on tank diameter) of 0.024, 0.048 and 0.072 cm/sec.

The Karim et al. (2004) geometry and operating conditions were duplicated to perform the 3D CFD simulations. Since the density of the slurry used in the experiments was approximately equal to that of water, the liquid phase was simulated with the physical properties of water. The gas phase was simulated with air. 3D steady-state simulations were carried out using CFD software CFX-5.5. $k-\varepsilon$ model was used to model the turbulence for liquid phase and zero equation model for gas phase (Sokolichin et al., 2004). Only drag force contribution was considered for the interphase force terms; drag force was modeled with Grace correlation.

After a thorough simulation of conditions and design used by Karim et al. (2004), different geometries of the digester were simulated by varying the diameter of draft tube, clearance of draft tube from bottom of the tank, and shape of the tank bottom. Table 1 lists the CFD simulations performed with different geometries and operating conditions.

Figures 1b and 1c shows the geometry of the conical bottoms used in simulations 7, 8 and 9 (refer to Table 1). The liquid height in simulations 7, 8 and 9 was maintained at...
23 cm to eliminate liquid volume as a variable; and ensure that the mixing energy provided in each configuration as power input per unit volume was constant.

**Results**

Figure 2a shows the velocity vector plot obtained from 3D CFD simulation of a flat bottom digester (simulation 1, Table 1) and compares it to CARPT results in Figure 2b. The main features of the flow pattern are revealed in Figure 2. Two distinct circulation loops are formed, one bigger loop which circulates the liquid into the draft tube (riser) from the bottom and out at the top of the draft tube and another smaller circulation loop, which exists near the liquid surface at the top of the tank. It can also be observed from Figure 2, that the maximum liquid velocities exist inside the draft tube. Very low velocities are found outside the draft tube (downcomer) and nearly zero velocities near the bottom of the tank.

The regions with very low velocities are termed stagnant or dead zones. The volume of dead zones can be found by calculating the total volume of the cells with very low liquid velocities. The regions with liquid velocities less than 5% of the maximum velocity were considered to be stagnant or inactive regions. For the simulation no.1 (refer Table 1) dead or stagnant volume in the digester equaled 59.7% of total digester volume.

Figure 3a shows the gas holdup distribution in the digester; the gas is only present inside the draft tube with very low gas hold up values, whereas gas holdup is zero near the wall regions. Since the gas sparger used was a single-point sparger, thus the gas distribution was non-uniform inside the draft tube. Figure 3a also shows the channeling and the non-uniform gas distribution with more gas near sparger pipe wall and less gas

<table>
<thead>
<tr>
<th>Simulation No.</th>
<th>Bottom</th>
<th>Draft tube diameter (cm)</th>
<th>Clearance of draft tube (cm)</th>
<th>Gas superficial velocity (cm/s)</th>
<th>% Dead volume</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Flat</td>
<td>4.4 (D/T) = 0.21</td>
<td>4</td>
<td>0.024</td>
<td>59.7</td>
</tr>
<tr>
<td>2</td>
<td>Flat</td>
<td>4.4</td>
<td>4</td>
<td>0.048</td>
<td>56.9</td>
</tr>
<tr>
<td>3</td>
<td>Flat</td>
<td>4.4</td>
<td>4</td>
<td>0.072</td>
<td>58.3</td>
</tr>
<tr>
<td>4</td>
<td>Flat</td>
<td>10.16(D/T) = 0.21</td>
<td>4</td>
<td>0.024</td>
<td>43.0</td>
</tr>
<tr>
<td>5</td>
<td>Flat</td>
<td>14.22(D/T) = 0.21</td>
<td>4</td>
<td>0.024</td>
<td>25.0</td>
</tr>
<tr>
<td>6</td>
<td>Flat</td>
<td>4.4</td>
<td>2 (draft tube height = 18 cm)</td>
<td>0.024</td>
<td>59.7</td>
</tr>
<tr>
<td>7</td>
<td>Conical (25°)</td>
<td>4.4</td>
<td>4</td>
<td>0.024</td>
<td>17.39</td>
</tr>
<tr>
<td>8</td>
<td>Conical (60°)</td>
<td>4.4</td>
<td>4</td>
<td>0.024</td>
<td>29.57</td>
</tr>
<tr>
<td>9</td>
<td>Conical (25°)</td>
<td>14.22</td>
<td>4</td>
<td>0.024</td>
<td>11.02</td>
</tr>
</tbody>
</table>

Figure 2 (a) Three dimensional CFD simulation results, (simulation no. 1); and (b) velocity vector plot from CARPT experiment data (Subscribers to the online version of *Water Science and Technology* can access the colour version of this figure from [http://www.iwaponline.com/wst/](http://www.iwaponline.com/wst/))
towards the wall of the draft tube. Figure 3a also agrees with the visual observation and the results obtained by CT measurements (Karim et al., 2004).

Figure 3b shows the solids holdup distribution. The solids concentration is higher at the bottom near the wall than the center because of the location of dead zones near the bottom wall. The solids concentration is uniform at the center of the digester due to higher velocities. The solids concentration is higher at the center at the surface of the slurry due to agglomeration. The CT scans shows that the solids hold up is not uniform throughout the digester.

Figures 2a and 2b shows good agreement of CARPT and CFD results qualitatively in terms of flow pattern, location of stagnant zones, and circulation loops. The volume of stagnant or inactive zones calculated from CARPT data is 58.8%, which is comparable to 59.7% as predicted by CFD.

Figure 4 provides the quantitative comparison of axial velocity radial profiles at different locations in the digester obtained by CFD and CARPT. The trend of velocity profiles obtained from CFD agrees reasonably well with the data obtained from CARPT.

Effect of air flow rate

The effect of air flow rate on the flow pattern was studied at three different gas flow rates mentioned earlier. The flow patterns obtained for three different gas flow rates were similar to the one shown in Figure 2. Figure 4b compares the axial liquid velocity profiles at the center of the digester for different air flow rates. Although the liquid velocities are
different inside the draft tube for different gas flow rates, not much difference in the velocity was seen in the region outside the draft tube. These results indicate that the flow pattern is not significantly affected by changing gas flow rate.

Effect of draft tube diameter
As shown in Table 1, the digester was simulated with three different draft tube diameters (simulations 1, 4 and 7). The ratio of draft tube diameter to tank diameter was varied from 0.2 to 0.5 and 0.7. Simulation results show that the flow pattern does not change with the draft tube diameter but the region over which liquid circulates increases with increasing draft tube diameter. Therefore the volume of dead or inactive zones decreases with increasing draft tube diameter.

Effect of draft tube height and clearance from the bottom
Draft tube height was increased by 4 cm (simulation 6) in order to keep the same clearance of draft tube from the bottom and from the liquid surface. The clearance was changed to 2 cm. No considerable effect of draft tube height or clearance was seen either on flow pattern, velocities or volume of dead zones.

Effect of the shape of the digester bottom
As shown in Figures 1b and 1c, two configurations of conical bottom were used to simulate their flow patterns using CFD. One bottom had a slope of 25° and other a slope of 60° (simulations 7 and 8, Table 1). The flow pattern obtained in these configurations looks similar to the one observed in Figure 2 for the flat bottom digester. However, the dead or inactive zones are reduced in the conical bottom (Table 1). The magnitude of the axial liquid velocity varies in the bottom region of the digester; the circulation velocities below the draft tube are lower for a conical bottom as compared to a flat bottom. The magnitude of velocity is similar in the upper region of the digester irrespective of shape of the bottom.

No appreciable difference in flow pattern or velocities was observed between 25° and 60° bottom. For convenience in construction the lesser sloped conical bottom (i.e. 25°) is preferred over steeper (i.e. 60°), as steeper cones are expensive to fabricate. Another reason to use the 25° bottom is that Choi et al. (1996) have reported lesser settling with 60° bottoms.

Simulation number 9 was carried out with the 25° angle conical bottom and draft tube diameter to tank diameter ratio (D/T) of 0.7, to see the combined effect of conical bottom and larger draft tube diameter.

Adding a conical bottom made no appreciable difference in either the flow pattern or the liquid velocity profile as compared to the flat bottom digester with D/T of 0.7. Figure 5 compares the axial liquid velocity radial profile at 2 cm from the bottom of the digester for 4 different configurations (simulations 1, 5, 7 and 9, Table 1). The effect of the larger riser diameter is more pronounced than the effect of the conical bottom; that explains why the velocity profile for the flat bottom overlaps the velocity profile for 25° conical bottom, both with D/T of 0.7. In sum, the difference in velocity profiles is not very significant due to change in bottom shape as compared to the change due to the increased D/T ratio.

Dead volume
The different configurations used for digesters in this study can be compared conveniently in terms of dead or inactive volume. Table 1 shows the dead zone volumes for different digester configurations. The gas flow rate has no effect on the dead volume,
whereas dead volume decreases with increasing draft tube diameter. Changing the D/T ratio from 0.21 to 0.7 decreases the dead volume by 60%. Again, the change in draft tube height and clearance has no effect on dead volume. The bottom with 25° slope has the least dead volume. The difference between dead volumes of 25° slope conical bottom with D/T of 0.21 and 0.7 is not significant.

**Conclusions**

The CFD predictions showed very good qualitative comparison with the experimental data in terms of flow pattern, location of dead zones, trends of velocity profiles and reasonable predictions of liquid velocity. The simulation results with a single-point sparger showed no effect of the gas flow rate in enhancing the liquid flow pattern. This result could be due to non-uniform gas distribution or low gas flow rates. More simulations must be performed to understand the effect of gas flow rate.

The increase in draft tube diameter results in higher liquid velocities outside the draft tube and also results in lower dead space volume. A conical bottom also helps in reducing the dead space. Interestingly, the effect of increasing riser diameter is more pronounced than changing the shape of the bottom. Thus, additive effect in mixing performance was observed by combining a change in the bottom shape with the largest diameter riser. Thus it can be concluded that the large draft tube diameter and/or a conical bottom should be used for practical reasons to enhance the digester’s mixing and thus the overall performance.

**Acknowledgements**

The US Department of Energy (DOE) is gratefully acknowledged for providing financial assistance in the form of a grant. The CFX software provided by ANSYS is also gratefully acknowledged.

**References**


