Jurnal Kejuruteraan 31(2) 2019: 201-208
https://doi.org/10.17576/jkukm-2019-31(2)-03

Analysis of Fluid Flow Patterns in Cylindrical Vessels of Anaerobic Digester using CFD

Muhammad Shakaib*, Antash Najib[b] & M. Ehtesham ul Haque[c]
[a]*Mechanical Engineering Department, NED University of Engineering and Technology*
Karachi, Pakistan
[b]*Mechanical and Aerospace Engineering Department, University of California, Davis, USA*
[a]*Mechanical Engineering Department, NED University of Engineering and Technology*
Karachi, Pakistan
**Corresponding author: mshakaib@neduet.edu.pk*

Received 12 August 2018, Received in revised form 30 November 2018
Accepted 26 March 2019, Available online 31 October 2019

ABSTRACT

The production of energy from biomass and waste material is gaining popularity worldwide due to the expected depletion of fossil fuels shortly. A device commonly used is anaerobic digester in which microorganisms react with biomass/biodegradable in a vessel and produce useful gases. The performance of anaerobic digester depends on mixing or proper contact of bio-matter with microorganisms. In this paper, flow behaviour is studied in cylindrical vessels of anaerobic digester using Computational Fluid Dynamics (CFD) technique. The study includes simulations for various cylinder dimensions obtained by changing the height and diameter of the digester vessel and locations for fluid inlet and outlet. The results are shown in qualitative terms using velocity profiles and quantitatively in terms of volume of the stagnant zone. The comparison of several geometries at a constant velocity indicates the considerable effect of cylinder aspect ratio (height to diameter ratio) and positions and numbers of inlet and outlet ports on mixing performance. The influence of inlet velocity/Reynolds number is also examined for a few cases. The digester which has an inlet and two outlets on the curved surface (both on the same side) is found to be most suitable. The volume of the dead zone in this configuration at various Reynolds number is less than 30% (based on 0.5% criterion).

Keywords: Anaerobic digestion; CFD; Reynolds number; stagnant zone

INTRODUCTION

Fossil-type fuels like petroleum, coal and natural gas are predominantly used and are the major source of energy. The advantages are that these fuels produce energy in huge amounts, are easy to transport and are common since the industrial revolution. A limitation, on the other hand is that the fossil fuels are non-renewable; the reserves are likely to be depleted within few decades. In addition, these fuels are largest source of greenhouse gas emission and hence one of the main reason for the change of climate. Keeping in view the limitations, stringent policies related to use of fossil fuels and greenhouse emissions are being made worldwide. Several countries are now decreasing reliance on fossils fuels and are planning to increase the use of alternative technologies to fulfill the energy requirements.

The biomass matter is abundant, easily available, has high carbon content with low ash and is renewable (Islam & Ani 2000). A process through which biomass can be converted into energy is anaerobic digestion. In this process, organic material such as manure, sewage, and other waste item is treated through microorganism activity in an oxygen-deficient system. The output from the process is biogas which is combustible and mainly contains methane and carbon dioxide. The biogas formed can be burned directly in a boiler to provide process or space heat, or it can be used to run a reciprocating engine to produce shaft/electric power. The AD process results in controlled release of gases which reduces the emission of greenhouse gases. Another benefit is that the process through treatment of a wide range of organic waste produces natural fertilizer which is a valuable by-product.

Use of anaerobic digestion process has rapidly increased globally and it is expected that it will be a dominant renewable energy source in future (Appels et al. 2011). Significant research has also been carried out to improve the process efficiency of this process in the recent decades. Among the important factors that affect the efficiency of AD are the residence time of slurry in the digester and the level of mixing of the incoming biomass material with the bacterial matter.

Mixing in the biodigester is desirable as it results in uniform distribution of microorganisms and substrate thus preventing stratification. Further, due to mixing, the particle size gets reduced which allows more release of gas from the mixture. The factors affecting the mixing performance of the anaerobic digesters have been investigated in numerous studies. For examining various flow processes CFD technique has now become feasible due to the advent of high speed digital computers (Othman et al. 2018). Several studies thus have investigated the process details using this technique. Monteith and Stephenson (1981) tested several digester geometries and showed that due to improper mixing, the effective volume reduces up to 30% of the total digester volume. Computer simulations by Bello-Mendoza and Sharratt (1998) revealed that partial mixing results in decreased methane production and inefficient treatment of waste mass.

Lopez et al. (2015) performed a 3D numerical study to examine the flow characteristics in an anaerobic digester of the Ontinyent Wastewater Treatment Plant. Pena et al. (2003) tested pilot-scale anaerobic ponds operating on domestic waste. The fluid flow structure and system performance were determined for various flow rates in the range 1.0-2.0 L/s. The findings indicated that the configurations containing vertical or horizontal baffles or mixing pit have favorable flow patterns and best efficiencies. Wu (2011) modeled turbulent flow of shear thinning fluids in anaerobic digesters. The research showed that a certain design of impeller with 15° spacing and inclinations 30° and 5° left and down respectively gives the optimum mixing results. It was further found that increase in total solids result in less efficient mixing.

Wu in other papers (2010; 2012) showed that there exists an optimal location for placing propeller which depends on the dimensions and geometry of the digester vessel. For example, in an egg-shaped digester with a working capacity of 4888 m^3, the suitable height of the propeller was mentioned to be 0.914 m below the liquid surface. It was also demonstrated that if the stirring power is increased, the amount of methane produced does not change significantly but energy efficiency decreases. Numerical simulations were performed by Wu and Chen (2008) in pilot and relatively large sized anaerobic digesters to determine the pumping power required for feed flow. The research work showed that for a digester of cylindrical shape and capacity of 1 m^3 and inlet pipe radius of 0.1R (R = radius of digester), the power input of 0.28 kW is sufficient. CFD simulations were conducted by Shen et al. (2013) to investigate the effect of impeller design and speed on biogas yield from rice straw.

The triple impeller with pitched blade at a speed of 80 rpm was mentioned to be the best combination. The doctoral thesis of Vesvikar (2006) showed that the draft tube diameter and the sparger geometry are two important parameters affecting the hydrodynamics of gas-lift digesters. Leonzio et al. (2018) compared various configurations and suggested that the fluid flow entrance tangent to the lateral surface of the anaerobic digester results in better mixing. Latha et al. (2009) performed CFD simulations using multiphase model for evaluation of mixing in an aerobic digester. In the transient case study, it was shown that the reactor achieves steady-state condition within 10 seconds of real time.

The review of literature shows that sufficient research work has been conducted to model fluid flow in anaerobic digesters in the previous years. However, improvements are still being proposed by the researchers to produce bio-digesters with enhanced hydrodynamic performance. The current research therefore includes CFD analysis for the effect of digester geometry that results in minimal stagnant zones and better mixing of fluid.

MODELING PROCEDURE

For numerical analysis, geometry is constructed and meshed into small control volumes in pre-processing software Gambit. The computational domain contains a cylindrical vessel with inlet and outlet pipes attached at the curved or circular top surface of the digester vessel. The volume for all the geometries is kept 500 m^3. Only half portion is created and modeled due to the symmetrical shapes considered. The effect of aspect ratio that is the ratio of height and diameter and locations of inlet and outlet pipe is examined. Three different aspect ratios are tested 1, 1.25 and 0.75 named as AD1, AD2, and AD3 respectively. The digesters with multiple inlet or outlet pipes are named as AD4-AD16. All the considered cases are shown in Figure 1.

The locations of inlet and outlet are either at the curved vertical surface at a distance of 5%, 50% or 95% (of total height 'H') from the bottom or at the top circular surface. The diameters of inlet and outlet pipes are equal to 0.4 m. Further details of the geometries are provided in Table 1.

TABLE 1. Digester geometries considered in this work

Name/Type	Description of Anaerobic Digester Geometry		
	Location of inlet(s)/Distance from bottom	Location of outlet(s)/Distance from bottom	Aspect Ratio (H/D)
AD1	0.05H	0.95H, opposite side	1
AD2	0.05H	0.95H, opposite side	1.25
AD3	0.05H	0.95H, opposite side	0.75
AD4	0.95H	Two outlets, at distances of 0.05H and 0.5H both on opposite side	1
AD5	Two inlets, at distances of 0.05H and 0.5H	0.95H, opposite side	1
AD6	0.5H	Two outlets, at distances of 0.05H and 0.95H on opposite side	1
AD7	Two inlets, at distances of 0.05H and 0.95H	0.5H, opposite side	1
AD8	0.05H	Two outlets, at 0.5H on same side, 0.95H on opposite side	1
AD9	0.5H	Two outlets, at 0.05H on same side, 0.95H on opposite side	1
AD10	0.5H	Two outlets, at distances of 0.05H and 0.95H on same side	1
AD11	Two inlets, at distances of 0.05H and 0.95H	0.5H, same side	1
AD12	Top surface	Two outlets, 0.05H on both sides	1
AD13	Two inlets, 0.05H on both sides	Top surface	1
AD14	Top surface	Two outlets, 0.05H on one side, 0.5H on other side	1
AD15	0.5H	Two outlets, top surface and 0.05H on opposite side	1
AD16	Two inlets, 0.5H and 0.05H on opposite side	Top surface	1

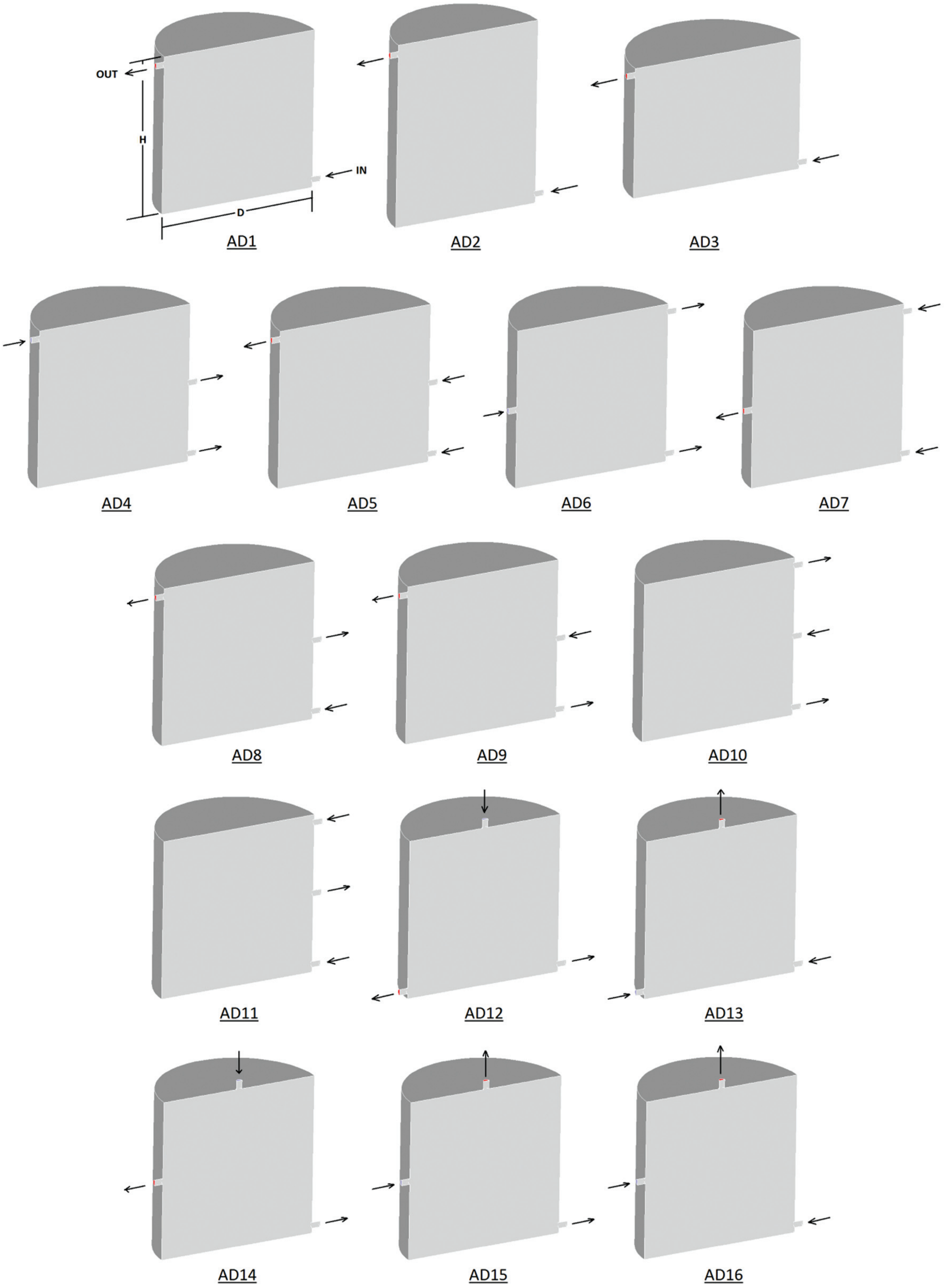

FIGURE 1. Anaerobic digester geometries considered

The computational domain is divided into number of cells as shown in Figure 2. The grid contains about 140,000 tetrahedral cells which are found to provide grid-independent results. Fluid flow in the anaerobic digesters is often non-Newtonian. However if the total suspended solids is less than 2.5%, the flow can be assumed Newtonian (Wu 2010). Fluid is thus assumed to be of constant viscosity and density. The governing equations are continuity and momentum equations for three- dimensional flow in the biodigester. The discretization of convective terms in the momentum

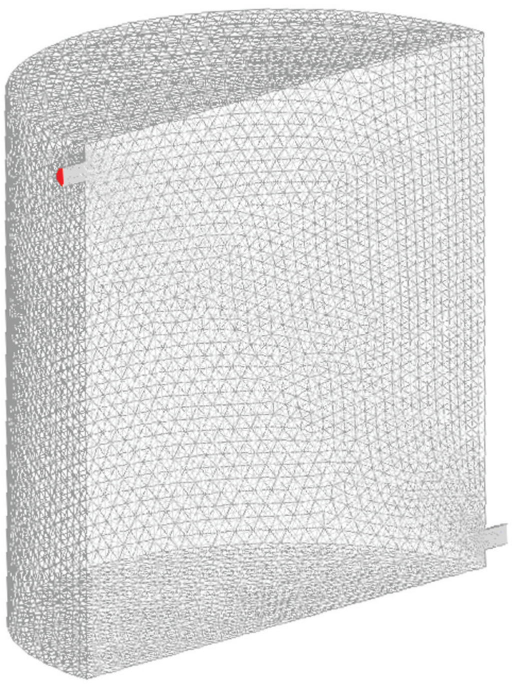

FIGURE 2. Computational grid for CFD analysis

equation is through second order upwind scheme while SIMPLE algorithm is used for coupling the pressure and velocity fields.

Convergence criteria are 1×10^{-5} for residuals of continuity and velocity components. The solution converges in about 1500 iterations as shown in Figure 3. The digester geometries are initially compared at fixed Reynolds number based on inlet pipe diameter of 200. Few digesters which yield better performance in terms of mixing/flow distribution are tested at other Reynolds number.

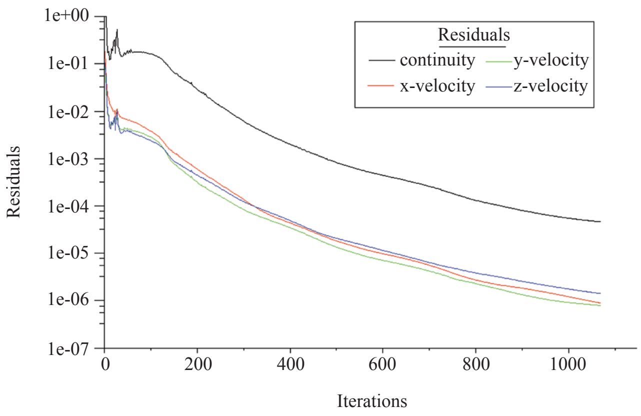

FIGURE 3. Residuals of continuity and velocity components versus number of iterations

RESULTS AND DISCUSSION

The velocity profiles in digesters with different aspect ratios are shown in Figure 4. The profiles indicate that velocity is higher in the bottom portion from where the flow enters and near the exit location in the top region for the three cases. In other regions of the vessel, the velocity magnitudes are relatively lower. As fluid flows towards the outlet, a major portion of fluid reverses which leads to a large recirculation region as clearly shown from the velocity vectors. The flow recirculation leads to mixing of the incoming fluid with the returning one. The velocity fields in the three digesters AD1-3 results show the presence of flow dead/stagnant zone of larger size. For example, in the top right corner, local velocities are almost zero and recirculation effect is also weak. The low velocity zone is found to be larger in digester AD2 which has a greater height.

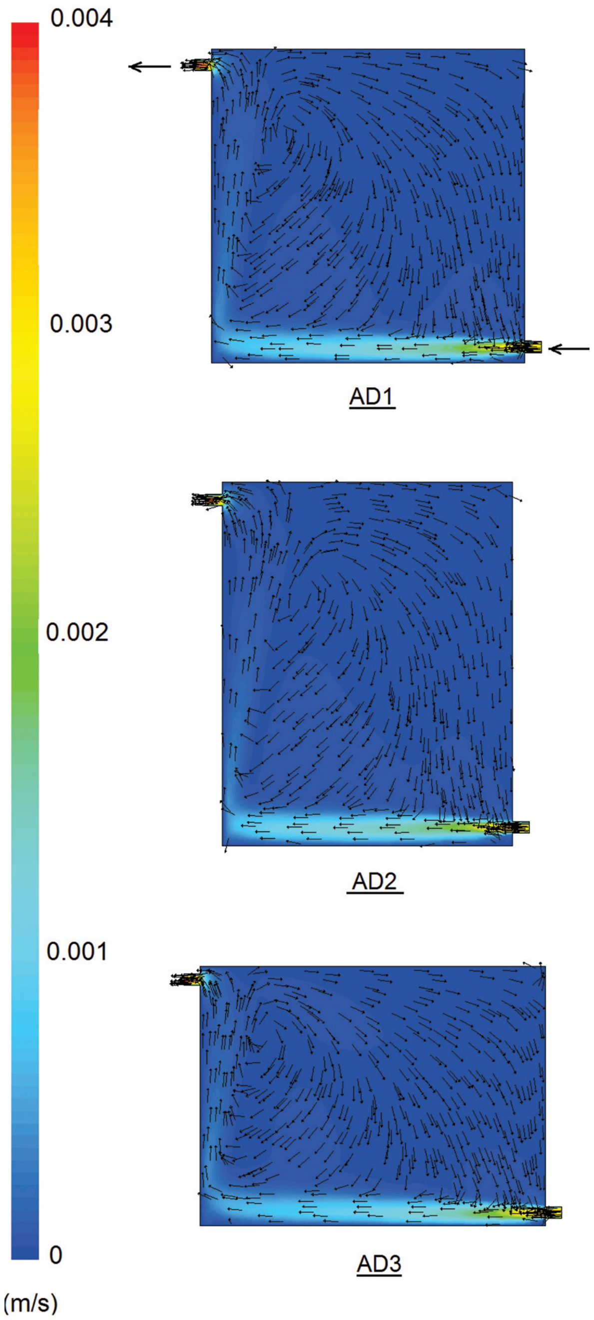

FIGURE 4. Velocity contours in digesters with different aspect ratios

In order to get improved performance in terms of mixing, geometries with multiple inlets or outlets are considered and contours are shown in Figure 5. Digester AD4 contains a single inlet and two outlets while AD5 has two inlets and one outlet on the opposite side. Even though AD4 has two outlets, the velocity profile is similar to AD1; a large recirculation vortex is formed as the fluid separates near the exit. In AD5, multiple flow recirculation regions are seen in the bottom region where two inlet velocity streams mix. Flow inlet from the middle and outlets from the opposite sides in upper and lower locations (AD6) or inlet from upper and lower sides and exit from opposite middle port (AD7) results in two symmetric recirculation regions. The velocity contours in various other geometries (AD8-AD16) also show that overall flow behavior and location of low velocity regions in digester depends on the inlet and outlet locations. Better distribution can be observed in AD10, AD11 which contains inlet and outlet pipes on the same side and AD15 as the local velocities within the vessel are relatively higher when compared to the other cases.

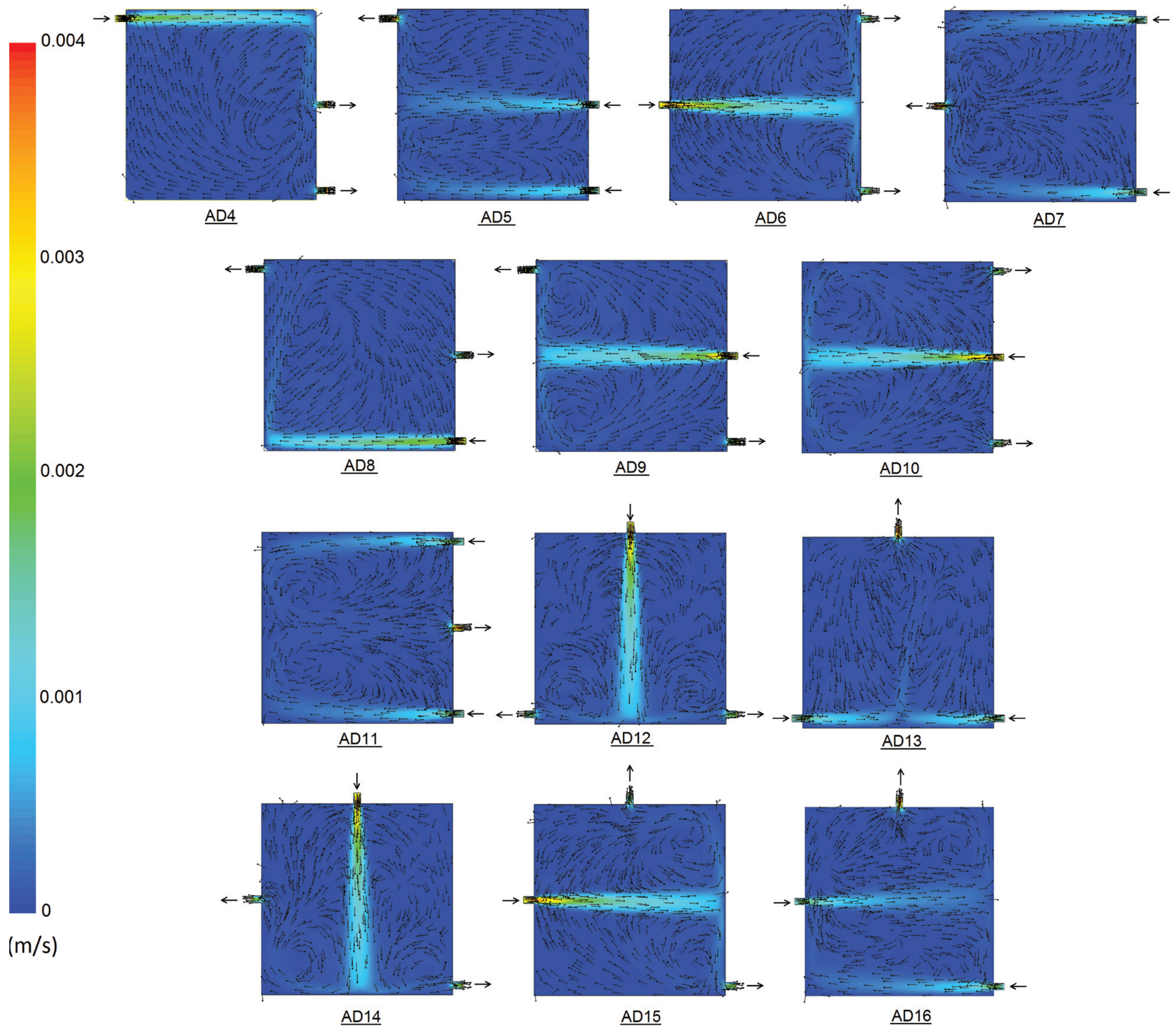

FIGURE 5. Effect of inlet and outlet positions on flow patterns

The anaerobic digesters are compared quantitatively on the basis of fraction of volume of dead zone (low velocity region). The volume of dead/stagnant zone is determined based on three criteria (i) 0.5, (ii) 1.0 or (iii) 5% of the inlet velocity (v_{in}). For example if v_{in} is 0.0025 m/s, the region which contains velocities below 1.25×10^{-5}, 2.5×10^{-5} and 1.25×10^{-4} m/s are termed 'dead zone' based on 0.5, 1.0 and 5% criteria. A lower value of this volume fraction is desirable. A comparison is given in Table 2. At 0.5% criterion, lower volumes of dead zone are found in geometries AD1 (22.9%), AD10 (23.8%) and AD2 (25.1%). When criterion is 1.0%, AD10, AD9 and AD1 have better results whereas at 5%, AD9, AD14 AD10 and AD12 yield lower volumes of stagnant zones. The values of stagnant volume fractions are higher in AD5, AD7 and AD16 which means less mixing. These types may therefore not be suitable for the anaerobic digestion process. The results shown in Figure 4 and 5 are for a fixed inlet Reynolds number of 200. The effect of Reynolds number (Re) or inlet velocity is studied for few cases in which better mixing performance is found at Re = 200.

TABLE 2. Volume fraction of stagnant region in various digesters

Digester Name	Volume fraction of stagnant zone		
	$v < 0.005v_{in}$	$v < 0.01v_{in}$	$v < 0.05v_{in}$
AD2	25.1	53.8	95.3
AD3	25.9	51.0	94.9
AD4	28.0	59.7	96.0
AD5	42.3	65.4	96.1
AD6	31.6	52.0	95.0
AD7	36.0	70.0	96.9
AD8	25.7	57.4	96.1
AD9	29.1	48.6	94.3
AD10	23.8	44.4	94.4
AD11	28.9	59.7	96.8
AD12	42.4	55.5	94.4
AD13	26.6	58.1	96.9
AD14	40.4	53.2	94.3
AD15	30.3	49.5	94.5
AD16	40.5	66.3	95.8

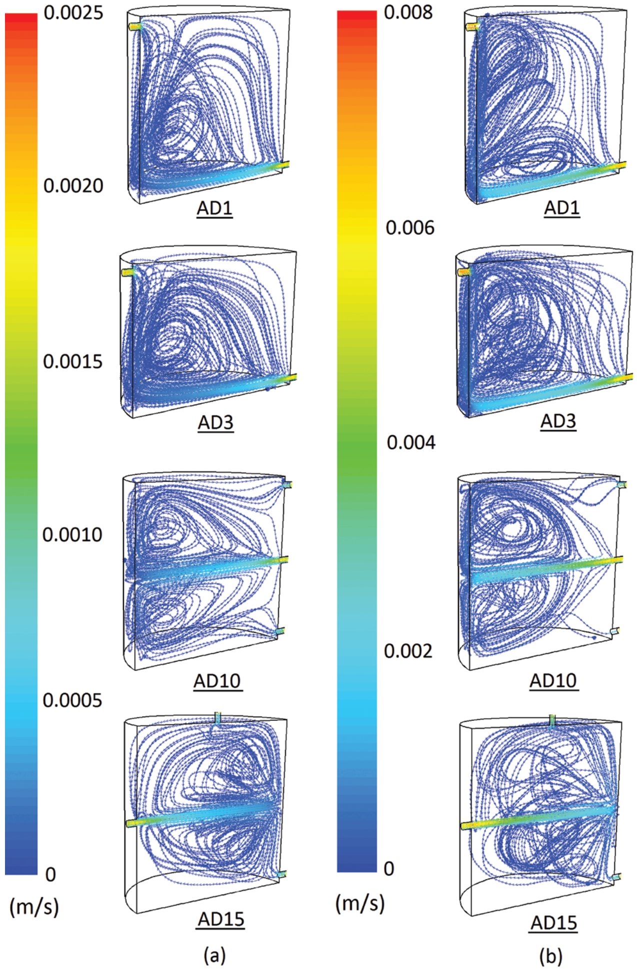

FIGURE 6. Path lines in AD1, AD3, AD10 and AD15 at Re = (a) 100 and (b) 400 respectively

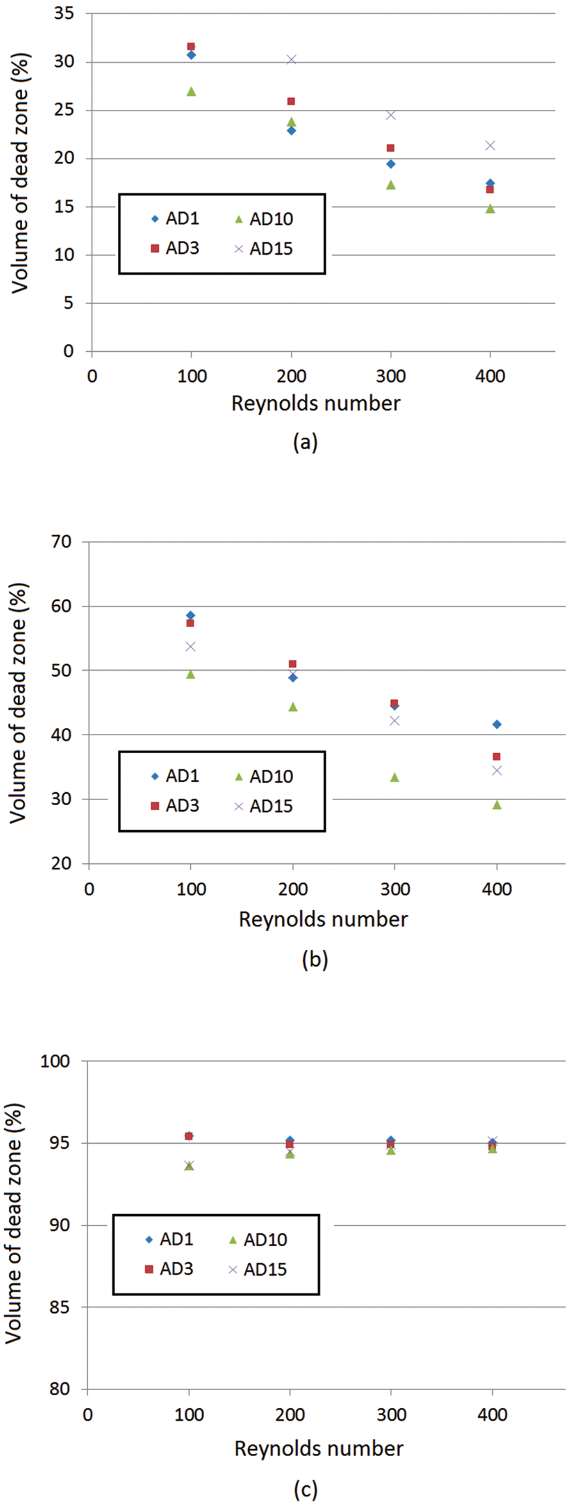

FIGURE 7. Volume fraction of dead zone with criterion (a) $v = 0.005v_{in}$ (b) $v = 0.01v_{in}$ (c) $v = 0.05v_{in}$

The fluid behaviors for the four cases are shown in Figure 6 with the help of path lines. At both Reynolds number, it is observed in digesters with single inlet and outlet (AD1 and AD3), the flow emerges from the lower (right) side, splits into two portions; a portion moves towards outlet (in the upper portion) while remaining recirculates and mixes with the entering fluid. This flow structure is similar to one which was seen in Figure 4 for AD1 and AD3. The fluid velocities in these cases are higher in the intermediate region (from inlet to outlet). The upper portion on the right side can be assumed stagnant zone as no path lines are seen. At higher Reynolds number, it is noticed that better flow mixing is achieved (particularly for AD3). In AD10 which has fluid inlet from the middle of the vertical side, two equal-sized and large recirculation regions are observed in the upper and lower portion at both Reynolds number. The fluid flow in AD15 after entering from left side is distributed; partially flows towards the top and partially towards the bottom to exit from the opposite lower side thus leading to recirculation zones of varying sizes. At higher Reynolds number, the path lines in AD15 appear to be random showing better mixing of the various flow regions. The volumes of dead zone at different Reynolds number are shown in Figure 7. The plot in Figure 7a which shows the volume of dead region based on 0.5% (of inlet velocity) criterion, indicate that AD10 results in minimum stagnant zones for most of the Reynolds number. Second suitable geometry is AD1 which has less dead volume when compared to AD3 and AD15. Similarly at criteria of 1 and 5%, AD10 has least value of dead volume. It is further noticed that the volume of dead zone decreases with the increase in Reynolds number particularly when criteria of 0.5 and 1.0% is used. For example, volume of dead zone decreases from 27% to about 15% in AD10 (at 0.5% criterion) when Reynolds number increases from 100 to 400.

The results of the present research work are compared with experimental results available in literature (Langner, 2009). Since the geometrical configurations considered in the present study are slightly different, for accurate comparison a cylindrical vessel with same dimensions to one used in Langner's study (G1, centre inlet) is created as shown in Figure 8. The simulation is carried out at same inlet velocity/ Reynolds number and flow patterns obtained are compared with the PIV (Particle Image Velocimetry) results.

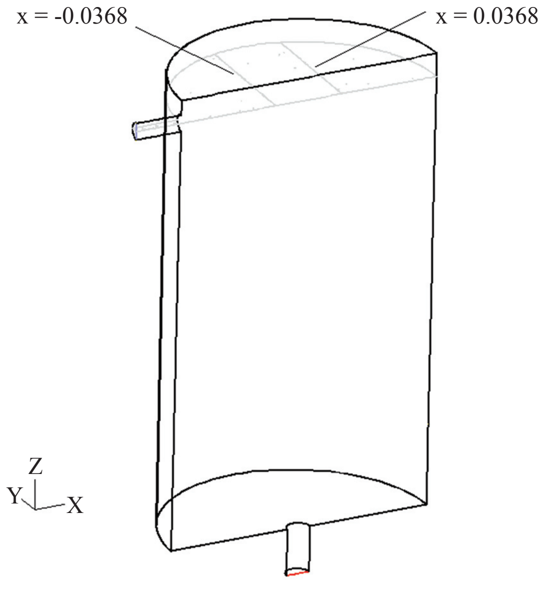

FIGURE 8. Domain for comparison with experimental results

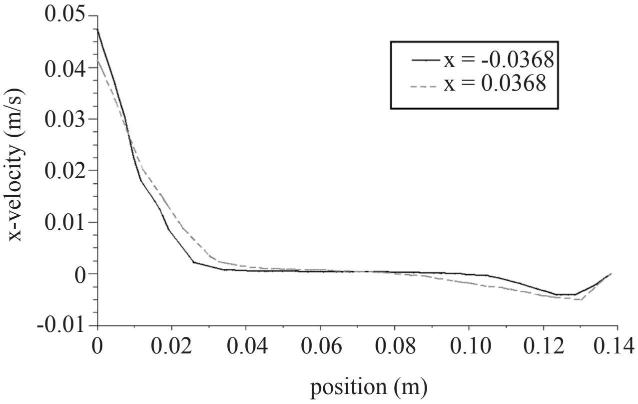

FIGURE 9. Velocity line plot at x = 0.0368 and x = -0.0368

The velocity contour from the present CFD study (not shown here) is found to be similar with the experimental results. A line plot of velocity profile at two x-positions is also obtained for comparison purpose and is shown in Figure 9. The line plot indicates that velocity is higher at position y = 0 (at the symmetric plane) because the fluid jet from the inlet pipe directly passes through this location with high velocity. Near the wall surface, velocity is found to be negative due to flow reversal. The comparison of this velocity plot obtained from CFD with the PIV results show that CFD results over-predict the maximum velocity values which occur at y = 0. The general trend is however found to be same and velocity values in the flow recirculation regions are found to be relatively closer. Thus the numerical results can be considered satisfactory.

The results of present study are also evaluated based on the grid-independence test by increasing the number of cells to 225,000 for digester AD1 and comparing the volume fraction values based on 0.5%, 1.0 and 5% criteria. The difference in terms of this volume is less than 1% which further shows reliability of this work.

CONCLUSIONS

CFD study in this paper predicts flow mixing behavior in anaerobic digestion vessels. The work shows presence of high velocity and flow recirculation regions in several regions of the digester. The fluid mixing is found to depend on the digester dimensions and locations of inlet and outlet ports. The geometry AD10 which contains two inlet ports on the side curved surface and outlet on the same side yields better results. The volume of dead region is found to be lowest for this geometry. The analyses at different Reynolds numbers showed that velocity distribution improves when Reynolds number is increased. Finally fair agreement is found of the present CFD results with the experimental ones in literature.

ACKNOWLEDGEMENTS

The authors acknowledge the support provided by NED University of Engineering and Technology, Karachi.

REFERENCES

Appels, L., Lauwers, J., Degrève, J., Helsen, L., Lievens, B., Willems, K., Impe, J.V. & Dewil, R. 2011. Anaerobic digestion in global bio-energy production: Potential and research challenges, *Renewable and Sustainable Energy Reviews* 15(9): 4295-4301.

Bello-Mendoza, R. & Sharratt, P.N. 1998. Modeling the effects of imperfect mixing on the performance of anaerobic reactors for sewage sludge treatment, *Journal of Chemical Technology & Biotechnology* 71(2): 121-130.

Islam, M.N. & Ani, F.N. 2000. Production of Bio-oil from Palm Oil Shell. *Jurnal Kejuruteraan* 12(2000): 105-116.

Langner, J.M. 2009. Investigation of non-Newtonian flow in anaerobic digesters. Thesis PhD, University of Manitoba, Canada.

Latha, S., Borman, D.J. & Sleigh, P.A. 2009. CFD multiphase modelling for evaluation of gas mixing in an anaerobic digester. *Proceedings of 14th European Biosolids and Organic Resources Conference and Exhibition*, 1-16.

Leonzio, G. 2018. Study of mixing systems and geometric configurations for anaerobic digesters using CFD analysis, *Renewable Energy* 123: 578-589

López-Jiménez, P.A., Escudero-Gonzáleza, J. Martínez, T.M., Montañana, V.F. & Gualtieri, C. 2015. Application of CFD methods to an anaerobic digester: The case of Ontinyent WWTP, Valencia, Spain. *Journal of Water Process Engineering* 7: 131-140.

Monteith, H.D. & Stephenson, J.P. 1981. Mixing efficiencies in full-scale anaerobic digesters by tracer methods, *Journal of the Water Pollution Control Federation* 53(1): 78-84.

Othman, N.A.T. & Farid, A.N.M. 2018. Droplets tracing in a T-junction microchannel. *Jurnal Kejuruteraan* 30(1) 2018: 47-53.

Pena, M.R., Mara, D.D. & Piguet, J.M. 2003. Improvement of mixing patterns in pilot-scale anaerobic pond treating domestic sewage. *Water Science and Technology* 48(2): 234-242.

Shen, F., Tian, L., Yuan, H., Pang, Y., Chen, S., Zou, D., Zhu, B., Liu, Y. & Li, X. 2013. Improving the mixing performances of rice straw anaerobic digestion for higher biogas production by computational fluid dynamics (CFD) simulation. *Applied Biochemistry and Biotechnology* 171(3): 626-642.

Vesvikar, M.S. 2006. Understanding the hydrodynamics and performance of anaerobic digesters. Thesis PhD, Washington University, USA.

Wu, B. 2010. CFD simulation of mixing in egg-shaped anaerobic digesters. *Water Research* 44(5): 1507-1519.

Wu, B. 2011. CFD investigation of turbulence models for mechanical agitation of non-Newtonian fluids in anaerobic digesters. *Water Research* 45(5): 2082-2094.

Wu, B. 2012. Integration of mixing, heat transfer, and biochemical reaction kinetics in anaerobic methane fermentation. *Biotechnology and Bioengineering* 109(11): 2864-2874.

Wu, B. & Chen, S. 2008. CFD Simulation of non-Newtonian fluid flow in anaerobic digesters. *Biotechnology and Bioengineering* 99(3): 700-711.