International Journal of Technology (2016) 2: 197-203
ISSN 2086-9614 

# CFD ANALYSIS OF SLURRY FLOW IN AN ANAEROBIC DIGESTER

Ahmad Indra Siswantara[1*], Asyari Daryus[1,2], Steven Darmawan[1,3], Gun Gun R. Gunadi[1,4], Rovida Camalia[1]

[1]*Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Kampus UI Depok, Depok 16424, Indonesia*
[2]*Department of Mechanical Engineering, Universitas Darma Persada, Jakarta Timur 13450, Indonesia*
[3]*Department of Mechanical Engineering, Universitas Tarumanagara, Jakarta 11440, Indonesia*
[4]*Department of Mechanical Engineering, Jakarta State Polytechnic, Depok, West Java, Indonesia*

(Received: October 2015 / Revised: December 2015 / Accepted: January 2016)

## ABSTRACT

This study uses biogas, an environmentally friendly renewable energy resource, to operate the prototype of a micro-gas turbine (MGT) system called the Proto X-3 Bioenergy Micro-gas Turbine, designed for green building application. The biogas is produced by an anaerobic digester. The aim of this research is to simulate slurry flow in an anaerobic digester as the basis for developing a biogas digester that will produce biogas to meet the requirements of the Proto X-3 Bioenergy Micro-gas Turbine. The digester is a rectangular type with 3.4 $m^3$ capacity. The flow calculations and simulations were done using Computational Fluid Dynamics (CFD) methods in two-dimensional, body-fitted coordinate mesh. The simulations were conducted with various baffle clearances for the digester: 50 mm, 100 mm, and 150 mm. The CFD simulations showed that the recirculation phenomena was found in all flows but that the 50-mm baffle clearance model had the largest recirculation, and it would lead to better mixing of the slurry.

*Keywords:* Anaerobic digestion; Biogas; CFD; Green building; Micro gas turbine; Turbulence modelling

## 1. INTRODUCTION

Anaerobic digestion is a process that uses micro-organisms to break down biodegradable materials into other organic and inorganic compounds without using oxygen (Craig et al., 2013; Mendoza et al., 2011). In anaerobic digestion, as a result of these anaerobic bacterial activities, the sludge will be transformed into a stable product and a gas, called biogas, which can be used as an energy source alternative to fossil fuel (Coughtrie et al., 2013; Lopez-Jimenez et al., 2015). A stable digester can be expected to produce approximately 50–60% methane, 30–40% carbon dioxide, 5–10% hydrogen, and small amounts of other substances by volume. This gas could be used for heating or for fuelling electric generators (Coughtrie et al., 2013).

The performance of an anaerobic digester is influenced by various factors, such as the degree of contact between the substrate and the bacteria, the mixing conditions, the temperature, the pH, the feeding patterns, the hydraulic retention times, and the substrate content (Coughtrie et al., 2013; Shen et al., 2013; Wu et al., 2009; Wu, 2014). The degree of contact between the substrate and the bacteria depends on the degree of mixing, and sufficient contact provides better conversion of the biomass to methane. Sufficient mixing prevents stream clogging and

*Corresponding author's email: a_indra@eng.ui.ac.id, Tel. +62-21-7270032, Fax. +62-21-7270033
Permalink/DOI: http://dx.doi.org/10.14716/ijtech.v7i2.2977

ensures good heat and mass transfer (Wu, 2014).

The flow in a digester is multiphasic, and modeling such a flow is more complicated than modeling a single-phase flow, increasing modeling costs. Karim, as quoted by Coughtrie et al. (2013), reduced the multiphase model to a single phase in experiments that found that the gas hold up was not significant in the main annular section of the digester, allowing gas phasing to be dispensed with. Coughtrie confirmed these results (Coughtrie et al., 2013).

Fluids can be classified as Newtonian and non-Newtonian. Lopez-Jimenez et al. (2015) conducted simulations for both types of fluids in an anaerobic digester, and the results showed that the velocities for both type of fluids, both upward and downward, and the percentage of dead zone were of the same order of magnitude for both types. Therefore, the assumptions regarding Newtonian fluids apply in a digester.

Because of the properties of slurry and the large size of an industrial-scale digester, the experimental methods used to analyze the flow phenomena in a digester are expensive and time-consuming. Therefore, the Computational Fluid Dynamics (CFD) method is useful, because it helps to describe the flow predictions in various digesters, reducing both expense and time. Over the last 20 years, many studies have used CFD to model flows in digesters, resulting in many beneficial inventions and theories that have contributed to low-cost, efficient digesters (Coughtrie et al., 2013; Shen et al., 2013).

The CFD calculations are influenced by the type of flow: laminar, transitional, or turbulent. Some experiments have shown laminar flow while others, at higher velocities, have shown transitional or turbulent flows. Wu, as quoted by Craig (Craig et al., 2013), stressed that most digester flows are turbulent and used a turbulence model to analyze the flow phenomena using CFD. The most appropriate model is the Large Eddy Simulation (LES), but it is expensive, whereas the Reynolds Averaged Navier-Stokes (RANS) models are low-cost but less precise. The Shear Stress Transport (SST)$k$-$\omega$ of RANS models provides the best results in a digester regarding gas mixing and agitating non-Newtonian slurry (Craig et al., 2013). Lopez-Jimenez et al. used STD $k$-$\varepsilon$ to analyze the flow patterns in a 2380 $m^3$ anaerobic digester to find the best design (Lopez-Jimenez et al., 2015), and (Mendoza et al., 2011) also used STD $k$-$\varepsilon$ to model digester flow.

This study evaluates a laboratory-scale anaerobic digester producing biogas used to drive a micro-gas turbine (MGT). Such flow simulation is necessary before constructing the actual digester to reduce the costs of design and construction. The various designs are simulated to find the one that is optimal, especially regarding slurry flow. In the digester used for this study, the slurry flows in one direction, so the simulation could be modeled in two dimensions, reducing the complexity of the calculations.

An MGT system is a gas-turbine system that produces energy, usually less than 500 kW (Paepe et al., 2014; Renzi et al., 2014). The biogas produced by the experimental digester will be used by an MGT, the Bioenergy Proto X-3, designed for green building application and capable of running on various types of fuel, especially renewable energy sources such as biogas, bioethanol, and biodiesel.

The aim of this study was to find the optimum design of flow-circulation in an anaerobic digester using CFD simulation with various baffle clearances.

## 2. METHODOLOGY

### 2.1. Geometry

Figure 1 shows the type of digester used, a laboratory-scale digester consisting of a main tank with baffles to maintain the flow, a screw pump to feed the biomass, and storage to collect the stable product. The gas is released from a pipe on top of the digester. The length of the main tank is 4 m, the width is 1 m, and the height is 0.65 m, not including the roof. The tank's internal volume is about 3.4 $m^3$. The inlet duct dimension is 0.1×0.2 $m^2$.

Mixing in a digester can be done using impellers, sludge pumping, and gas (Lopez-Jimenez et al., 2015). In this study, the pump, located in the front of the digester to feed the biomass, also acted as a pusher to push the sludge to the right, making the mixing occur.

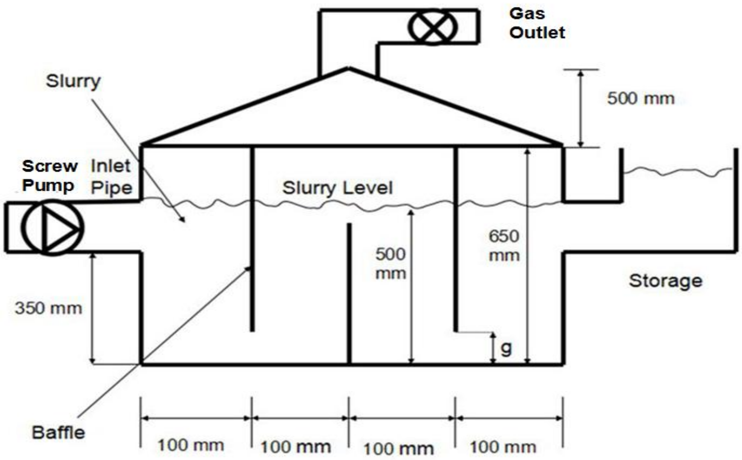



Figure 1 Anaerobic digester geometry (side view), g = Baffle Clearance

## 2.2. Governing Equations

The CFD simulation solves the basic conservation laws of fluid mechanics, conservation of mass, momentum, and energy in it's calculations.

The mass conservation equation is:

$$\frac{\partial \rho}{\partial t} + div(\rho \mathbf{u}) = 0 \quad (1)$$

where $\rho$ is the fluid density, **u** is its velocity vector, and $t$ is time.

The momentum equation in the x-direction is:

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + div(\rho u \mathbf{u}) = -\frac{\partial p}{\partial x} + div(\mu \,\text{grad}\, u) + S_{Mx} \quad (2)$$

where $p$ is the static pressure, μ is viscosity, and $S_M$ is the momentum source.

## 2.3. Physical Parameters and Boundary Conditions

Steady state conditions were used for simulations, and the fluid was assumed to be Newtonian and single phase. The density of the slurry was 999.66 kg/$m^3$, and the dynamic viscosity was 0.065 Pa.s. The flow modeling focused on sludge movement to find the optimal degree of mixing of the fluid. The velocity at the inlet was assumed to be 0.025 m/s. The flow in the inlet duct and the digester were laminar.

## 2.4. CFD Simulation

The CFD calculations were done using a commercial CFD software application, CFDSOF®, which performed pre-processing, including geometry creation, mesh creation, and boundary type setup, processing, including CFD condition setup and calculating the results, and post-processing, which included presenting the results. Figure 2 shows the CFD model. The grid was created in two dimensions and was a Cartesian type with 4600 cells (Figure 3).

The simulations were done with various baffle clearances, or gaps, to find the optimum flow mix for the slurry. Three baffle clearances were used: 50 mm, 100 mm, and 150 mm. The results were analyzed only for the velocity parameter, since the mixing conditions could be interpreted from this parameter. The velocity profiles were read at a distance of 40 mm and 470 mm from the base of the digester to get a wider view of the mixing conditions in the area (Figure 2).

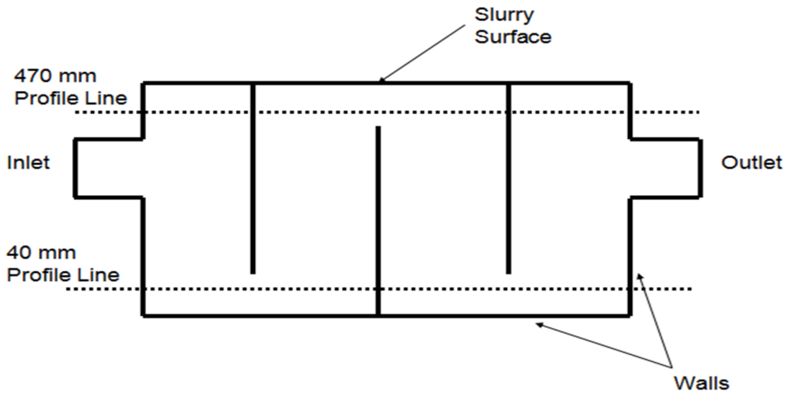



Figure 2 CFD model geometry

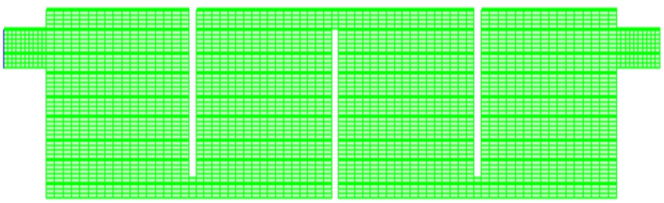


Figure 3 Meshed geometry

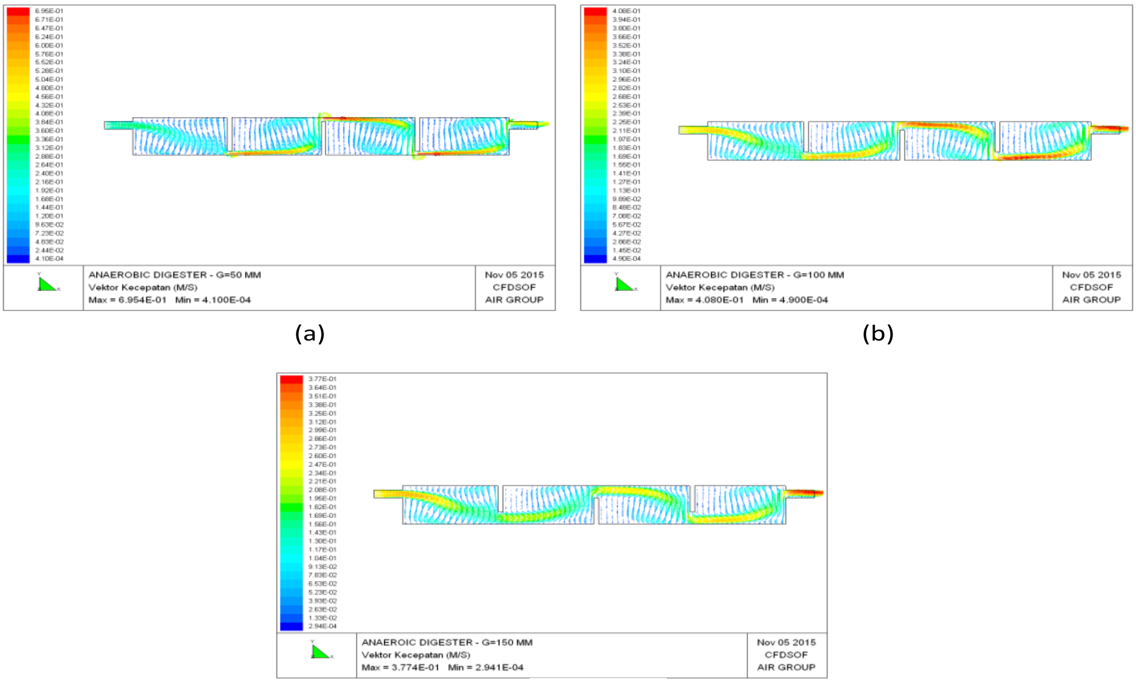
## 3. RESULTS AND DISCUSSION

Figure 4 shows the velocity vectors for the three baffle clearance conditions. The highest velocity, 69.5 cm/s at the baffle clearance, was achieved at 50 mm baffle clearance (Figure 4a). The next highest velocity, 40.8 cm/s at the outlet and the baffle clearance, was achieved at 100 mm baffle clearance (Figure 4b). A velocity of 37.7 cm/s was achieved at the outlet at 150 mm baffle clearance (Figure 4c).

As Figure 5 shows, recirculation was found in every compartment in all simulations. However, the maximum recirculation was seen with the 50-mm baffle clearance (Figure 5a), followed by the 100-mm and 150-mm baffle clearances, respectively (Figures 5b and 5c). Since the 50-mm clearance provided the highest velocity, a higher velocity may provide greater recirculation.

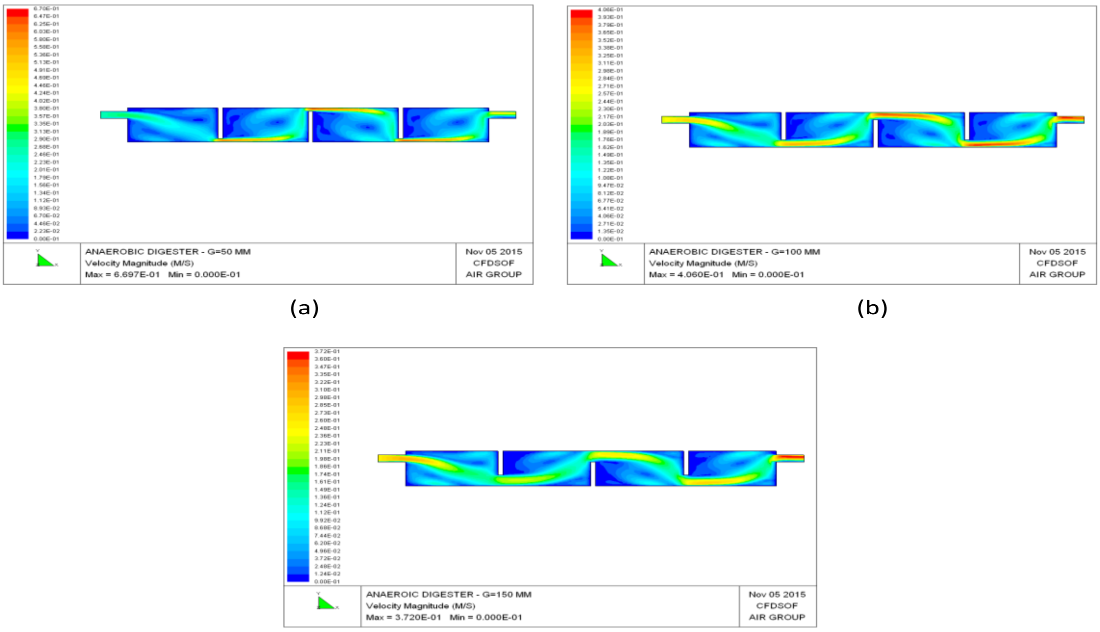
The velocity profiles can be used to predict velocity magnitude and the location where the recirculation will occur. The highest velocity is predicted to result in the largest recirculation and to lead to better mixing. Optimal mixing produces the maximum amount of biogas.

Figure 6 shows the velocity profiles for three baffle clearances near the bottom of the digester, 30 mm from the base. The recirculation might appear at distances of 0.25 m, 2.25 m, and 2.75 m from the left side of the digester, as indicated by the increased velocities at these points. These three simulations provided results that were most similar at 0.25 m and 2.25 m, unless the 50-mm baffle clearance had a higher velocity at 2.75 m, in which case that distance might provide better mixing. The peak of the curve is the velocity at the baffle clearance.



(c)

Figure 4 Velocity vector for the liquid phase in the horizontal length: (a) 50 mm; (b) 100 mm; and (c) 150 mm of baffle clearance



(c)

Figure 5 Contour of velocity magnitude for the liquid phase: (a) 50 mm; (b) 100 mm; and (c) 150 mm of baffle clearance

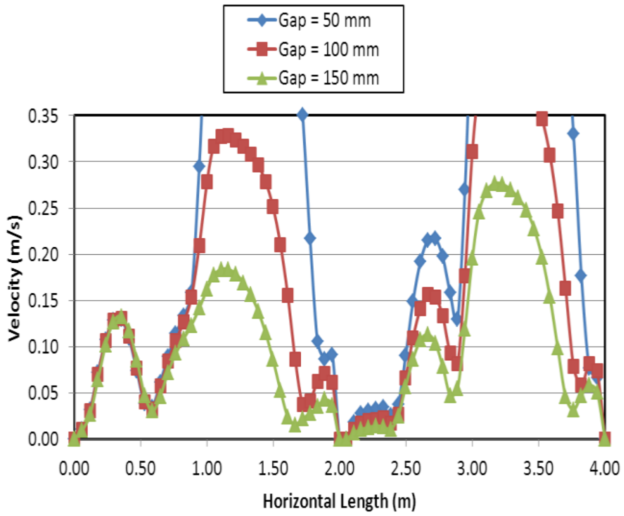



Figure 6 Velocity profiles at 30 mm from the base of the digester

Furthermore, Figure 7 shows the velocity profiles at a distance of 470 mm from the base, or 30 mm from the top of the slurry level. Similar to the velocity profile for the bottom of the digester, this profile showed velocity increases at 0.75 m, 1.75 m, and 2.75 m from the left side of the digester. The 50 mm baffle clearance also showed the highest velocity of the three, indicating that it might provide more recirculation and thus more homogeneity to the slurry mix. It might be concluded that the 50 mm baffle clearance provided better mixing than the other two clearance distances.

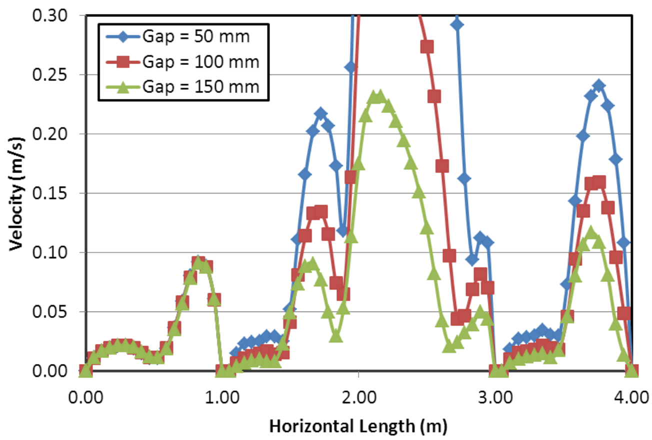



Figure 7 Velocity profiles at 470 mm from the base of the digester

## 4. CONCLUSIONS

The mixing processes inside the digester, which are influenced by the flow phenomena, determine the quantity and quality of the products. To investigate these flow phenomena inside the digester, CFD simulation was conducted on a rectangular, anaerobic digester with laminar flow at three different baffle clearances. The following conclusions can be drawn from the study: (1) the highest velocity in the compartments, 69.5 cm/s, was found at the 50 mm baffle clearance distance, which was predicted to create larger recirculation in the flow; (2) the 50 mm baffle clearance distance also provided the maximum recirculation; (3) the 50 mm baffle clearance distance provided the best results among the three baffle clearance distances; (4) to increase the homogeneity of mixing, further studies should investigate the another mixing method, such as mixing using sludge pumping, impellers, or gas.

## 5. ACKNOWLEDGEMENT

The authors would like to thank the Ministry of Research, Technology, and Higher Education of the Republic of Indonesia, who supported this research through the Hibah Insentif SINas 2015, and PT. CCIT Group Indonesia for providing the CFDSOF® software license.

## 6. REFERENCES

Coughtrie, A.R., Borman, D.J., Sleigh, P.A., 2013. Effects of Turbulence Modelling on Prediction of Flow Characteristics in a Bench-scale Anaerobic Gas-lift Digester. *Journal of Bioresource Technology*, Volume 138, pp. 297–306

Craig, K.J., Nieuwoudt, M.N., Niemand, L.J., 2013. CFD Simulation of Anaerobic Digester with Variable Sewage Sludge Rheology. *Journal of Water Research*, Volume 47, pp. 4485–4497

Lopez-Jimenez, P.A., Escudero-Gonzalez, J., Martinez, T.M., 2015. Application of CFD Methods to an Anaerobic Digester: The Case of Ontinyent WWTP, Valencia, Spain. *Journal of Water Process Engineering*, Volume 7, pp. 131–140

Mendoza, A.M., Martinez, T.M., Montanana, V.F., Jimenez, P.A.L., 2011. Modelling Flow Inside an Anaerobic Digester by CFD Techniques. *International Journal of Energy and Environtment*, Volume 2(6), pp. 963–974

Paepe, W.D., Contino, F., Delattin, F., Bram, S., 2014. Optimal Waste Heat Recovery in Micro Gas Turbine Cycles through Liquid Water Injection. *Journal of Applied Thermal Engineering*, Volume 70, pp. 846–856

Renzi, M., Caresana, F., Pelagalli, L., Comodi, G., 2014. Enhancing Micro Gas Turbine Performance through Fogging Technique: Experimental Analysis. *Journal of Applied Energy*, Volume 135, pp. 165–173

Shen, F., Tian, L., Yuan, H., Pang, Y., Chen, S., Zou, D., Li, X., 2013. Improving the Mixing Performances of Rice Straw Anaerobic Digestion for Higher Biogas Production by Computational Fluid Dynamics (CFD) Simulation. *Journal of Applied Biochemistry and Biotechnology*, Volume 171, pp. 626–642

Wu, B., Bibeau, E.L., Gebremedhin, K.G., 2009. Three-dimensional Numerical Simulation Model of Biogas Production for Anaerobic Digesters. *Journal of Canadian Biosystems Engineering*, Volume 51, pp. 8.1–8.7

Wu, B., 2014. CFD Simulation of Gas Mixing in Anaerobic Digesters. *Journal of Computers and Electronics in Agriculture*, Volume 109, pp. 278–286