Manipal Journal of Science and Technology

Volume 3 | Issue 2 Article 1

12-31-2018

CFD Modelling to Evaluate the Effect of Digester Temperature on Biogas Production

Shuichi Torii
Kumamoto University, Kurokami, Kumamoto, torii@kumamoto-u.ac.jp

Research Articles

CFD Modelling to Evaluate the Effect of Digester Temperature on Biogas Production

Huqe Farhana and Shuichi Torii*

Email: torii@kumamoto-u.ac.jp

Abstract

Biogas production from chicken manure and food solid waste has grabbed the attention of engineers and managers globally because of the substantial advantages in achieving environmental protection, generation of energy and Green House Gas emission reductions. However, there are a number of problems involved in scaling up experimental Anaerobic Digestion (AD) plants to field level plants. One of the major problems associated with AD is mixing, which is a key component in segregating synthesized gas and biomass from digester liquid, enhancing homogeneity and in ensuring adequate contact between bacteria and substrate in the Anaerobic Digestion. Besides this, temperature plays a vital role in enhancing the action of microbial species in order to initiate and continue the digestion process. Sometimes it requires increasing temperature from mesophilic to thermophilic condition for obtaining a higher concentration of methane in raw biogas. But, as the digester is an opaque system, it is difficult to design and maintain correct temperature range in the reactor. Such situations are well suited to the Computational Fluid Dynamic (CFD) analysis, where models can be utilized to realize the rheological behaviour of the sludge at the suitable temperature range (including lower and higher temperature) in terms of vector field pattern of the slurry. To achieve such an intent, Reynold's Averaged Nevier-Stokes equation and k-ε turbulence model was used in CFD simulation. The goal of this paper is to understand the rheological pattern of variation of the velocity of the slurry particles with temperature. Besides, this study reveals the initial and stable temperature condition in enhancing biogas production, which can be applied in the near future in both laboratory scales and industrial scales of biogas production in order to curtail energy crisis in the modern civilization.

Keywords: Anaerobic digestion, computational fluid dynamics (CFD), turbulence models, vector field pattern, residuals

I. Introduction

In recent years, Anaerobic Digestion (AD) technology has been developed increasingly by environmental engineers and scientists to generate biogas and power from organic solid waste as a source of clean and green energy. AD technology not only generates biogas as an energy source, but it also reduces the emission of green house gases, and it has become a mandatory requirement for developed and developing countries in the management of solid wastes. Biogas is an important renewable energy source for rural areas in developing countries. It is produced by Anaerobic Digestion of biological wastes. It is an environment friendly, clean, cheap, and versatile fuel. Biogas generally comprises 55-65 % methane, 35-45 % carbon dioxide, 0.5-1.0 % hydrogen sulfide and traces of water vapour. The average calorific value of biogas is 20 MJ/m3. The use of biogas systems can increase agricultural productivity. Biogas can be produced through various methods. Both combustion and digestion are onvenient processes for its production. Anaerobic Digestion of solid wastes in the absence of oxygen is the easiest for the production of biogas. This is because this technique can be handled with a low cost and it can even be conducted in both a small and

Huqe Farhana[1], Shuichi Torii[2]

[1]*Doctoral student, Department of Mechanical System Engineering, Kumamoto University, Kurokami, 2-39-1, Kumamoto*

[2]*Professor, Department of Mechanical System Engineering, Kumamoto University, Kurokami, 2-39-1, Kumamoto*

Manuscript received: 1 Nov 2018
Revision accepted: 28 Nov 2018

* Corresponding Author

How to cite this article: Huqe Farhana and Shuichi Torii "CFD Modelling to Evaluate the Effect of Digester Temperature on Biogas Production", Manipal J. Sci. Tech., vol.3(2), 1-11, 2018.

Published by Impressions@MAHE, 2018

a medium scale industry. But, a change in the feed to the digester can cause various problems in the Anaerobic Digestion, particularly with the mixing. A lot of research has been carried out on the Anaerobic Digestion, and good process monitoring can help, control, optimize, and evaluate the biogas process very well. In particular, a new challenge is to optimize the mixing systems in biogas digesters to increase the biogas production rate. Scum, foam, and froth are major causes of concern in Anaerobic Digesters as they have the potential to cause a failure of the digester operation (in part due to their prevention of gas release). Thus, the gas, liquid, and solid (GLS) separation in an AD is vital for the success of the technology for any solid waste AD. Efficient mixing has been proved to increase the biogas production many folds enabling the further recovery of power from the solid waste. Biogas mixing has been reported to be less expensive and easier to operate than the impeller and slurry recirculated mixing [16], but it has not yet been optimized to maximize the biogas yield. In the future reactor design, a high solid loading is necessary to reduce the size of reactor units while maintaining a relatively low capital investment. High rate Anaerobic Digesters of high Organic Loading Rates (OLR) and short hydraulic retention times have become an attractive codigesting option for AD in the recent years. However, high solid loading contributes to the problem of mixing inefficiency, and more energy is required to complete the mixing. Previously, a research article suggested that minimal mixing may improve high solids Anaerobic Digestion by providing quiescent environmental conditions for bacteria [8]. For proper Anaerobic Digestion, apart from these criteria, the action of bacteria is controlled by another vital factor; temperature. For biogas production, there are certain ranges of temperature in which the microbial community starts its biochemical reactions. Depending upon the properties of the waste materials, the most favorable temperature conditions are: mesophilic condition (from 25oC to 30oC) and thermophilic condition (from 40oC to 50oC) [20]. But it is tough to maintain the correct temperature level as the environmental temperature can also affect the digestion temperature. In such a situation, Computational Fluid Dynamics (CFD) can be useful in understanding the rheological manure of the sludge in both lower and higher temperatures. Besides, the continuous change of the velocity of the particles of the sludge with the increase in temperature can also be observed by means of CFD simulation; so that the temperature condition can be optimized in a realistic Anaerobic Digester. This is the foremost goal of this study.

2. Literature Review

It has been a decade since biogas is being used as an alternative energy source in developed and developing countries, and the obstacles associated with biogas have become a major barrier in achieving the targeted objective of biogas technology. For this reason, these obstacles have opened a new window for the engineers to find out the appropriate solutions both virtually and practically. Specially, the application of CFD simulation has created a great possibility in realizing the mixing behaviour of the sludge computationally. A 3D model of a covered lagoon digester was developed to solve such a problem at the very beginning of this century [20]. This complex model incorporated the processes of bulk fluid motion, sedimentation, bubble mixing, bubble entrainment, advection, biological reactions, and heat transfer. The model was validated for using performance data from full scale digesters. The importance of mixing in achieving efficient substrate conversion has been reported by several researchers [10, 5, 18]. The main factors affecting digester mixing are the mixing intensity and duration, the location of the feed inlet and outlet and the type of mixing. However, the effect of mixing duration and intensity on the performance of Anaerobic Digesters are contradictory. For instance, adequate mixing has been shown to improve the distribution of substrates, enzymes and microorganism throughout the digester [7]. Mathematical modelling is also useful in understanding the Anaerobic Digestion; especially this type of modelling is helpful in obtaining the degradation phases [19]. All these CFD simulations contain the basic rheological variation of the waste sludge. Computational Fluid Dynamics (CFD) has become a popular tool for reactor analysis, because it allows the investigation of local conditions in an arbitrary vessel size, geometry and operating conditions [14]. CFD techniques are being increasingly used for experiments to obtain detailed flow fields for a wide range of fluid types. The capability of CFD tools in forecasting the mixing behaviour in terms of mixing time, power consumption, flow pattern and velocity profiles is considered as a successful achievement of these methods and acceptable results have been obtained in many applications [10]. Besides, CFD simulation can also be useful in understanding the rheological nature of the sludge. Several studies have been conducted to obtain the 3D and 2D pictures of the mixing behaviour of the liquid waste material [2, 17]. With the help of these results, the conditions for maximum biogas production can be optimized and reset. There are several factors that are highly responsible for biogas production. The effect of temperature inside the digester, the PH of the sludge, the mixing ratio between the solid wastes and water are the most important factors that can affect the biogas production. The role of temperature is a vital factor for the enhancement of biogas production [20, 1]. In this study, the effect of temperature shock in the digester has been focused by means of computational fluid dynamics (CFD) simulation. The Standard k-ε turbulent model has been used to solve the turbulent problem. Star CCM+ commercial software has been used to carry out the simulation process. The main goal of this study is to reveal the rheological variation of the velocity of the sludge particles with the increase of temperature. The variation of velocity of the sludge particles has been obtained as the vector field pattern. The magnitude of the velocity of the sludge particles has also been taken into account.

Objectives of this Study

Utilization of biogas as a substitution for fossil fuel is not a new idea. But it still has not been utilized in an extended field. It is well known to all that, biogas production by Anaerobic Digestion from solid wastes often takes longer time. This is a major obstacle in utilizing biogas in an extended field of application than any other renewable energy sources. According to some recent research mentioned in the literature review section of this paper, it has been observed that if the reactor temperature can be increased, biogas production can also be enhanced [20, 1, 12, 4]. Here lies the main objective of this paper. In this paper, the vector field pattern of velocity has been perceived to obtain the variation of this field contour with temperature, so that we can predict the digestion computationally in order to optimize the digestion conditions before setting the real Anaerobic Digester. The faster increment in velocity with temperature implies to the faster biochemical reaction rate will take place in the mimic reactor. And the faster reaction rate obviously implies that a faster biogas production. Besides, through CFD simulation, the temperature at which a steady state can be reached has also been determined so that we can predict the favourable temperature range for biogas production.

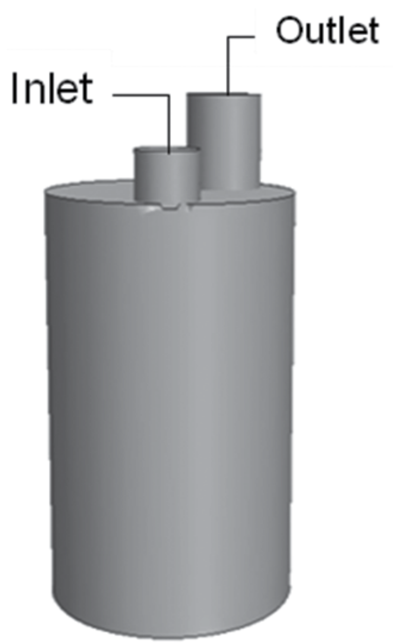
Development of the Computational Modelling

Mathematical models are being used to obtain the biochemical degradation pattern. Among them, Anaerobic Digestion Mode 1 (ADM1) is the most convenient mathematical model [11]. The modified ADM1 has been used to find the methane production from anaerobic digestion process [22]. But these are attempts that included the 2D modelling for biogas production. In contrast, Computational Fluid Dynamics (CFD) is useful in obtaining the 3D flow pattern, which is more useful to realize the digestion process for methane production. In this study, we have used Star CCM+ to conduct the CFD modelling. The main objective of our study is to observe how the velocity of the sludge particle changes with the temperature inside the anaerobic digester. This model solves, numerically, the laws governing the fluid dynamics, solving equations by a geometric domain. The basic flow equations used are the Navier-Stokes and the momentum equation in discrete form. The magnitudes of both velocity and temperature are calculated in a discrete manner at the nodes of a mesh or a network, describing the flow geometry modelling.

Building the schematic geometry of the mimic anaerobic digester

To obtain an acceptable simulated output, a symmetrical geometric structure has been built computationally. The geometry of the model was built on the basis of a plastic cylindrical biogas digester. The height of the digester was 30m and the diameter was 7.5m. The thickness of the plastic wall was 1.2m. The total capacity of the digester was 500 litres.

Maintaining the similarities with the real dimensions of the biogas digester, the replica was built using the AutoCAD 3D software, and then converted into the file type lithography in order to be implemented in the STAR CCM+ Software. For convenience of the simulation, the dimensions of the digester were minimized. The height of the digester was 0.65m and the diameter was considered to be 0.25m. The inlet pipe for the insertion of the raw material and the biogas outlet was also made with suitable dimensions. Figure 1 represents the schematic geometry of the digester.



]

Figure 1: Sketch of the geometry of the digester

Generation of mesh for simulation

Generation of the mesh is an important step for the CFD simulation. As the mesh is refined a more accurate solution is obtained. But the more refined base size of the cells chosen, the more time will be required to solve the turbulent problems. Besides, the selection of a proper mesh model is also crucial to have an appropriate meshing. For a satisfactory meshing, both surface meshing and volume meshing were taken into account. For surface mesher, surface remesher and surface wrapper remesher models were selected. For volume mesh, volume polyhedral mesher, advanced layer mesher and prism layer mesher models were designated. Before choosing the base size, an independent meshing was performed considering three sizes with 25% variation from the selected base size. But no significant change was obtained. To achieve acceptable meshing within the short computational time, the following parameters were chosen:

Table 1: Mesh parameters

Parameters	Values
Base size	0.0075 m
Surface curvature	Default
Surface growth rate	Default
Number of prism layers	2
Prism layer thickness	0.33 m
Prism layer stretching	1.5
Wrapper feature angel	30o

The base size was selected to be 0.075m, and this size was preferred on the basis of the shortest computational time. Figure 2 presents the sketch of the surface mesh and Figure 3 presents the sketch of the volume mesh.

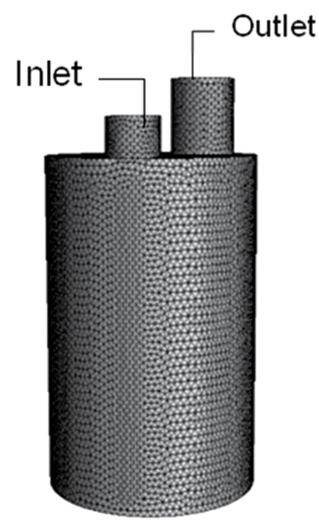


Figure 2: Sketch of the digester after surface mesh

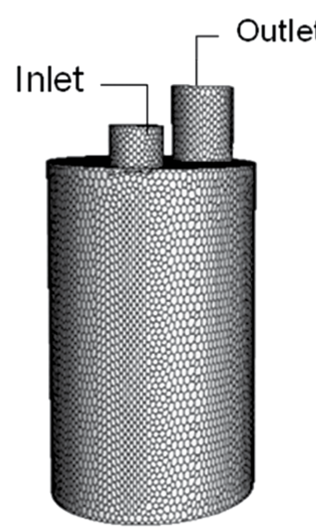


Figure 3: Sketch of the digester after volume mesh

The computational load was low, which generated 2,71,840 cells, 17,57,811 faces and 14,59,830 vertices after simulation. Besides, the convergence criteria were determined not to exceed 0.001 for continuity, mass, momentum, and energy.

Specifying the physics models for simulation

To conduct the CFD simulation, we used the commercial package Star CCM+. This software provides conventional physics models that can easily be used to define the properties of the 3D CAD model. In our work, a simplified case is taken where no flow in or out of the reactor is considered. This is a reasonable representation of the initial conditions in a real reactor at the start of a new process. As this process works in a stationary state and only liquids (water and activated sludge) were simulated, a 3D and segregated flow was selected. The fluid was assumed to be Newtonian and single phase. The flow modelling focused on the sludge movement to find the optimal degree of mixing the fluid. The goal of this study is to obtain the variation of the vector field of velocity with the increase of temperature. The physics conditions that are chosen in order to carry out the CFD simulation mentioned in table 2.

Table 2: Specification of the physics models

Properties	Properties
Space	Three dimensional
Time	Implicit unsteady
Material	Liquid
Equation of state	Constant density
Viscous regime	Turbulence
Reynolds-Averaged Turbu-lence	Standard k-ε
Wall treatment	All Y+(positive Y direc-tion)

Besides, the fluid was assumed to be Newtonian and single phased. Constant density liquid, which was the material simulated in the continuum, was responsible for managing the velocity and the physical processes being modeled in the continuum. Table 3 illustrates the properties of liquid used in the present study.

Table 3: Selected material properties

Properties	Values
Density	0.786 g/cm3
Dynamic viscosity	5.43×10^{4} Poise
Specific heat	2527.18 J/Kg-K
Turbulent Prandtl Number	0.85

Specification of initial and boundary conditions In our study, the anaerobic digester was separated into four parts to obtain suitable CFD simulation: inlet, outlet, boundary and wall. Here, the inlet was defined as a classical velocity inlet; the boundary was specified as a symmetric plane; the outlet was defined as a pressure outlet, and the sides of the digester were specified as wall. The values of the velocity were calculated based on mass balance. For the other parts of the digester, the values were computed based on thermal specification. Besides, for solving turbulence problems, turbulence specification was taken into account.

Elucidation of the turbulence modelling for simulation

There are several approaches to solve the turbulence problems in the CFD simulation. In the present study, the Reynolds Averaged Navier-Stokes equation was used. The flow inside the digester was considered to be turbulent, and for this reason, standard k-ε turbulence model was implemented. This is a mathematical technique which is mostly used in computational fluid dynamics to simulate the flow characteristics for turbulent flow conditions. A large number of iteration has been taken into account to obtain a suitable result from the simulation. The turbulent Length Scale that is used in this case is approximately 7% of the inlet length. The turbulent Velocity Scale that is used in this case is 10% of the inlet velocity, that is 1.0 msec-1. Turbulent intensity was 0.01.

The averagred transport equations With the concept of Reynolds time-averaging and the rules defining its application, we turn our attention to the general conservation equations governing the fluid flow and transport phenomena. The equation of continuity is given by,

Published by Impressions@MAHE, 2018

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (2)$$

Here and after, the tensor (index) notation is used such that repeated indices indicates summations. Taking the Reynolds time-average of equation (1) gives,

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (2)$$

Together with the equation of continuity, we need another equation to complete the CFD simulation, which is called momentum equation and the momentum equation is given by,

$$\frac{\partial(\rho U_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}[2\mu S_{ji}] + \rho g \quad (3)$$

Where

$$S_{ji} = \frac{1}{2}\left(\frac{\partial}{\partial x_j}(u_i) + \frac{\partial}{\partial x_i}(u_j)\right) \quad (4)$$

The Standard k-ε Turbulent model

The standard k-ε model is the mostly used model in computational fluid dynamics for turbulence flow. It is a two equation model which gives a general description of the turbulence-by means of two-transport equations-PDEs) [6].

The first transported variable is the turbulence kinetic energy (k) and the second transported variable is the rate of dissipation of turbulence energy (ε). The exact k-ε equations contain many unknown and immeasurable terms. For a much more practical approach, the standard k-ε turbulence model is used, which is based on our best understanding of the relevant processes, thus minimizing unknowns and presenting a set of equations that can be applied to a large number of turbulent applications. The turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations:

For kinetic energy k,

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (5)$$

And equation for dissipation ε,

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon$$

(6) The term Gk represents the generation of turbulence kinetic energy due to the mean velocity gradients and Gb is the generation of turbulence kinetic energy due to buoyancy. The term YM presents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. Besides, these equations consists of some adjustable constants, such as σ k, σ_ε, $C_{1\varepsilon}$ and $C_{2\varepsilon}$. The values of these constants have been arrived at by numerous iterations of data-fitting for a wide range of turbulent flows.

These are as follows:

Eij = component of rate of deformation

μ t= eddy viscosity

ρ= density

σ k= 1.00

σε= 1.30

C1ε= 1.44, C2ε= 1.92

Besides, the turbulent heat transport is modelled using the concept of Reynolds' analogy to turbulent momentum transfer. The "modelled" energy equation is given by the following expression,

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}[u_i(\rho E + \rho)] = \frac{\partial}{\partial x_j}\left(k_{eff}\frac{\partial T}{\partial x_j} + u_i(T_{ij})_{eff}\right) + S_h \quad (7)$$

Here, E is the total energy, K_{eff} is the effective thermal conductivity and τ_{ij} is the deviatoric stress tensor and the expression for $(\tau ij)_{eff}$ is as follows,

$$(T_{ij})_{eff} = \mu_{eff}\left(\frac{\partial \mu_j}{\partial x_i} + \frac{\partial \mu_i}{\partial x_j}\right) - \frac{2}{3}\mu_{eff}\frac{\partial \mu_i}{\partial x_i}\delta_{ij} \quad (8)$$

The term $(\tau ij)_{eff}$ represents viscous heating, and is always computed in the coupled solvers. The effective thermal conductivity can be expressed as follows,

$$k_{eff} = k + \frac{c_p \mu_t}{P_{r_t}} \quad (9)$$

Here k is the thermal conductivity and the value of Pr_t is 0.85. This value can be changed according to the situation. Now the turbulent viscosity can be computed by combining k and ε as follows:

$$\mu_t = \rho c_\mu \frac{k^2}{\varepsilon} \quad (10)$$

Here, Cμ is a constant. The model constants $C_{1\varepsilon}$, $C_{2\varepsilon}$, Cμ, σk and σε have the following default values:

$C_{1\varepsilon}$= 1.44, $C_{2\varepsilon}$= 1.92, C_μ= 0.09, σ_k= 1.0 and σ_ε= 1.3

These default values are standard and most widely used for the CFD simulation. But these values can be changed (if needed) from the "viscous panel" of the software.

Results and Discussion

From recent research articles, it has been obtained that, modern Computational Fluid Dynamics (CFD) software helps modellers to define mixing efficiencies for different digester configurations before construction; or to determine possible virtual changes in the construction parameters. CFD visualization and analysis also opens a window for the researchers to examine input configurations to design a real anaerobic digester. In computational analysis, there are some parameters which are very useful in characterizing the required design. Velocity is one of such parameters. This is because, the vector field pattern of velocity can easily describe the flow pattern. Additionally, vector field pattern is also useful to predict the mixing behaviour of the raw materials inside the opaque digester. In this paper, the main goal is set to observe how the vector field changes with the increment of reactor temperature. Besides, to obtain a suitable simulated output, a large number of iteration has been considered. There are eight consecutive velocity contours that were recorded to inspect the variation of velocity field. Figures 4(a-h) represent the velocity contour diagrams. These vector field patterns were recorded at eight chronological temperatures. The temperature scale was set between mesophilic temperature to thermophilic temperature,

which means; the initial temperature was chosen to be 298K (25°C) and the final temperature was specified to 333K (60°C). After completing the simulation, the lowest magnitude was obtained at 0.164m/sec and the highest magnitude of velocity was found to be at about 4.94m/sec. The blue coloured region presents the lower magnitude of velocity and the red coloured region denotes the higher velocity magnitude.

Figure 4(a) presents the velocity contour at 298K (25oC). From this picture it can be observed that- at this initial stage, the velocity of the sludge particle is very low. For this reason, most of the area of this diagram is covered by blue colour. Additionally, the presence of the red colour, which is initiated at the top of the diagram, denotes the increment of velocity magnitude with temperature. Then, figures 4(b) and 4(c) present the velocity contour at 303K (30oC) and 308K (35oC), respectively. It is well established that, to produce biogas from solid wastes, the favourable temperature range is started from 298K (25oC) to 308K (35oC), and this temperature range is known as the "mesophilic temperature" [21]. From figures 4(b) and 4(c), it can be noticed that- the velocity of the sludge particles is being increased with the accretion of temperature inside the digester. In figure 4(b), we can monitor that the red coloured region has started to increase from the top of the vessel and is moving towards the bottom at a clockwise direction. Furthermore, in figure 4(c), the centre of the vessel is wrapped by the red colour. These two figures denote the continuous development of velocity with temperature, which compliment the experimental results of biogas production [3, 10, 18, 20].

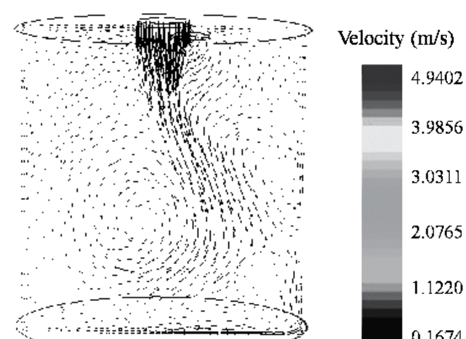


Figure 4(a): Velocity field contour at T=298K

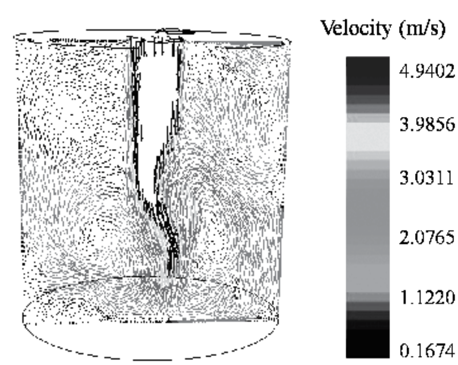


Figure 4(b): Velocity field contour at T=303K

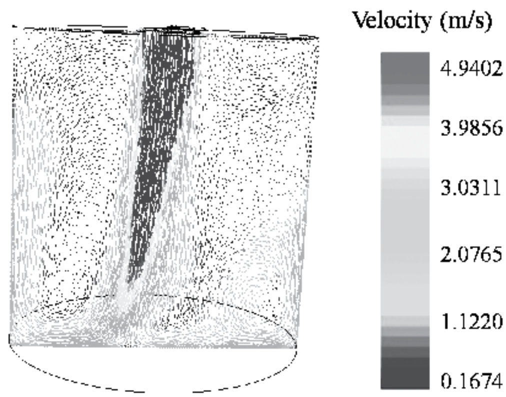


Figure 4(c): Velocity field contour at T=308K

The accession of velocity can also be observed being faster in figure 4(d). This diagram presents the velocity contour at 313K (40oC). From this velocity field contour, we can see that- centre of the diagram is covered with red and yellow colour and the bottom of the vessel is coated with a light blue colour. Such types of contours signify that the velocity of the sludge particles has been increased with a moderated magnitude existing in the whole vessel.

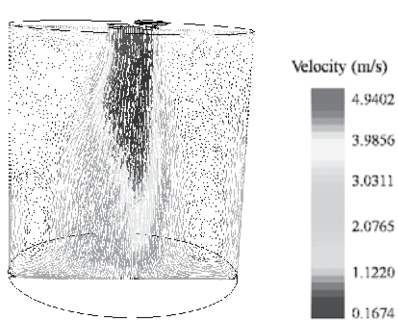


Figure 4(d): Velocity field contour at T=313K

Same velocity contour (figure 4(f)) is obtained after increasing the digester temperature to 318K (45oC). It means, that the predicament of velocity has not changed after getting an increment of more than 278K temperature. Even at 323K (50oC), the condition of the velocity field pattern has not changed (figure 4(g)). From these two situations we can say that- the steady state has been accomplished within 318K (45oC) and 323K (50oC).

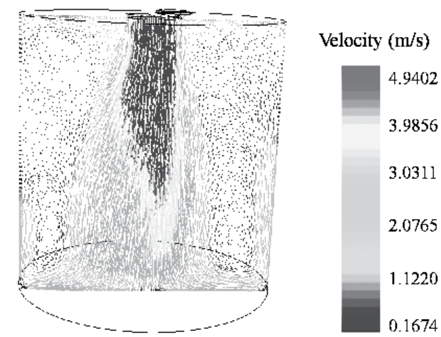


Figure 4(e): Velocity field contour at T=318K

After observing these two velocity field patterns, if we notice the velocity contour in figure 4(g), we can clearly see that, most of the area of this diagram is covered with a light blue colour and there is a small trace of the red colour. This signifies that- after getting an increment of more 278K (5oC) in the vessel, the velocity of the sludge particles has drastically decreased and in this situation it may be concluded that- the production of biogas has been decreased at temperature 328K (55oC).

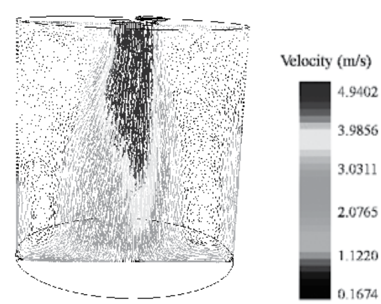


Figure 4(f): Velocity field contour at T=323K

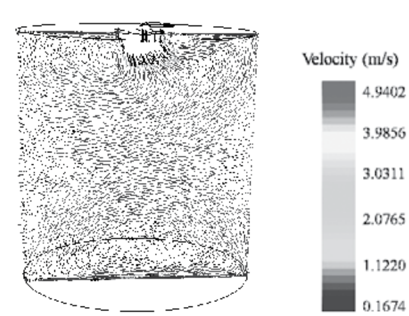


Figure 4(g): Velocity field contour at T=328K

Finally, from the velocity field pattern in figure 4(h), we can easily notice that- the velocity of the sludge particles has been lowered at temperature 333K (60oC) and this condition implies that- the biochemical reaction rate has been decreased at this temperature.

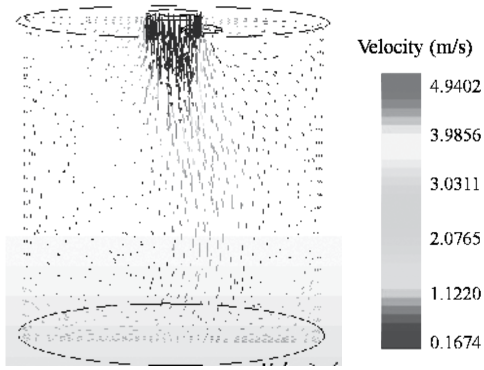


Figure 4(h): Velocity field contour at T=333K

Discussion

Anaerobic digestion is a very cost effective method for biogas production. But this technique is maintained by the action of some selective micro organisms, and the activities of these organisms are highly influenced by temperature. The main aim of this paper is to verify the influence of temperature on the velocity of the sludge particles by Computational Fluid Dynamics (CFD). Presently, many research works have revealed that, temperature plays an important role in enhancing the biochemical reaction for biogas production [3, 10, 18, 20]. The most favourable temperature range lies within 298K (25^{oC}) and 328K (55^{oC}). In our simulation, we obtained a similar result. We attained computationally that- the velocity of the sludge particles increases with the regular increment in the reactor. Besides, when the digester temperature reached 328K (55^{oC}), the velocity of the slurry particles started to decrease. Even when the reactor temperature increased to 333K (60^{oC}), the velocity is again lowered with the minimum magnitude. The experimental output of some researchers revealed that- after increasing the reactor temperature from 333K (60^{oC}) to 338K (65^{oC}), the biogas production decidedly decreased [3, 10]. They showed that- biogas produced faster both at thermophilic and mesophilic temperature. But after increasing the temperature beyond 328K (55^{oC}), the biogas production had started to reduce [8, 3]. These experimental outcomes are analogous with our simulated results.

Besides, a simulated outcome can be acceptable if the solutions of the simulation are converging. It is easy to verify the computational outputs by observing the pattern of the values of the residuals after the CFD simulation [15]. If the values of residuals are found to be continuous and stable after completing the simulation process, then the computational results are said to be converging. In accordance with this statement, the values of residuals are found to be stable after the entire process. Figure 5 presents the values of residuals after our simulation. From this figure it can be clearly observed that- the values of residuals are continuous and they are not at any descending manure.

Together, with this criterion, the base size of cells for meshing is also another vital matter, which is to have a coarse meshing for suitable simulation. For this reason, before selecting the best matched base size, three expanses were chosen with 25% variation. The measurement with minimum computational time was finally selected.

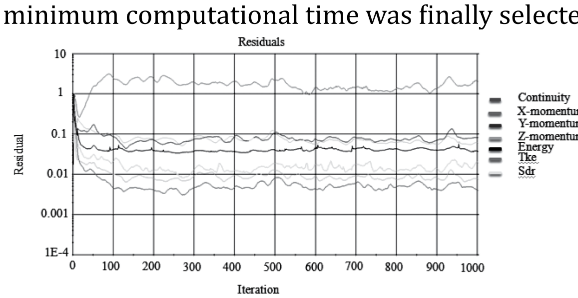


Figure 5: Values of residuals after simulation

Conclusion

In this paper, we have shown the application of CFD technique to obtain the optimized temperature conditions for maximum biogas production. The prime focus was to attain the variation of velocity of the sludge particles with the increment of digester temperature by means of vector field patterns. The following conclusions can be drawn after completing the simulation:

Researchers found that- temperature has a critical impact on the biogas production by anaerobic digestion. Besides, it is also revealed that- if the digester temperature can be increased, the biogas production process can be enhanced. For this reason, our main target was to establish this idea by inspecting the 3D computational diagram.

As meshing is the most influenced step to accomplish a suitable CFD simulation, the meshing models were selected properly. Besides, the base size was preferred to be either very small or very large. This measurement was checked by the skewness and aspect ratio on mesh metric and smoothness (change in cell size). Together with this, the parameters that are used to characterize the physics models were set properly. Last, but not the least, to obtain an acceptable solution, the simulation was carried out along with a large number of iteration.

To attain our result, the temperature range was scaled between mesophilc and thermophilic temperature (within 298K to 333K). During the simulation it was found that- the magnitude of the velocity of the sludge particles was being increased with temperature. From the vector field pattern of velocity, it can be observed that- the direction of the velocity was clockwise and the movement of the sludge particles was being started from the top and through the centre part of the vessel, it travelled the entire vessel. The minimum magnitude of the velocity was found to be 0.164 m/sec and the maximum magnitude was 4.9402 m/sec. The initial temperature was 298K (25oC) and the final temperature was 333K (60oC). The simulated results were found to be similar to recent experimental outcomes of some research groups that were discussed in this paper.

Monitoring the pattern of the values of the residuals during and after simulation is an important factor to clarify whether the solutions are acceptable or not. In our case, the values of residuals after the CFD simulation were found to be continuous and consistent. This is a fair confirmation that- the solutions of our simulation are converging.

In conjugation with the results of our simulation, we can conclude that- biogas production can be improved if the digester temperature can be increased with mesophilc temperature and thermophilic temperature conditions. But if the increment of the reactor temperature is beyond thermophilic condition, the biogas production may not take place as the liquids in the sludge may start to vaporize. In conjugation with the above conclusion, we can say that- anaerobic digestion can easily be optimized to recover biogas production in practical experiments, and it can help reduce the application of natural gas in our civilized life.

Acknowledgment

The authors would like to acknowledge the Ministry of Education, Culture, Sports, Science and Technology (MONBUKAGAKUSHO: MEXT for providing the financial support (PhD scholarship) for this research and the extended help of the Thermal engineering laboratory under the Department of Advanced Mechanical System Engineering, Kumamoto University, Japan, for providing the facility for the simulation process.

References

[1] Chae, K J et al (2008), The Effect of Digestion Temperature and Temperature Shock on The Biogas Yields from Mesophilic Anaerobic Digestion of Swine Manure, Bioresource Technology, vol. 99, pp. 1-6, DOI: 10.1016/j biortech.2006.11.063.

[2] Constaza, S R et al (2018), Computational Fluid Dynamics (CFD) Modelling in Anaerobic Digestion: General Application and Recent Advance, Critical Review in Environmental Science and Technology, Vol. 48, pp. 39-76, DOI: 10.1080/10643389.2018.1440853.

[3] Demollari, E et al (2017), Temperature and Stirring Effect on Biogas Production from Two Different Systems, American Journal of Energy Engineering, Vol. 5, No. 2, pp. 6-10, DOI: 10.11648/j.ajee.2017502.

[4] Ghatak M N and Mahanta, P (2014), Effect of Temperature on Biogas Production from Lignocellulosic Biomass, 1st International Conference on Non-Conventional Energy (ICONE-2014), 16-17 January, DOI: 10.1109/ ICONCE.2014.6808702.

[5] Kim, M et al (2001), Comparative ProcessStability and Efficiency of Anaerobic Digestion; Mesophilic vs. Thermophillic, Water Research, Vol. 36, pp. 4369-4385, PMID: 12420941.

[6] Launder, B E and Spalding, D B (1974), The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering, Vol. 3, No. 2, pp. 269-287, DOI: 10.1016/0045-7825.

[7] Lema, J M et al (1991), Chemical Reactor Engineering Concepts in Design and Operation of Anaerobic Treatment Processes, Water Sci. Technol. Vol.24, pp. 79-86, DOI: 10.2166/ wst.1991.0219.\

[8] Lettinga, G et al (1981), Feasibility of Anaerobic Digestion for the Direct Treatment of, and the

Energy Recovery from Urban Wastes, Study in Environmental Science, Vol. 9, pp. 97-109, DOI: 10.1016/S0166(08)71356-6.

[9] Mamun, M R A and Torii, S (2015), Anaerobic Co-digestion Technology in Solid Wastes Treatment for Biomethane Generation, International Journal of Sustainable Energy, Vol. 36, No. 5, pp. 462-472, DOI: 10.1080/14786451.2015.1043302.

[10] McMahon, K D et al (2001), Anaerobic Co-digestion of Municipal Solid Waste and Biosolids Under Various Mixing Conditions II Microbial population dynamics, Water Research, Vol. 35, No. 7, pp. 1817-1827, DOI: 10.1016/S0043-1354(00)00438-3.

[11]Ozgun, H (2018), Anaerobic Digestion Model NO.1(ADM1) for Mathematical Modeling of Full-Scale Sludge Digester Performance in Municipal Waste Water Treatment Plant, Biodegradation, pp. 1-10, DOI: 10.1007/s10532-018-9859-4.

[12] Pandey, P K and Soupir, M L (2012), Impacts of Temperature on Biogas Production in Dairy Manure Anaerobic Digestion, International Journal of Engineering And Technology, Vol. 4, pp. 629-631, DOI: 10.7763/IJET.2012.V4.448.

[13] Ramaraj, R and Unpaparom, Y (2016), Effect of Temperature on Performance of Biogas Production from Duckweed, Chemostry Research Journal, Vol. 1, No. 1, pp. 58-66, ISSN: 2455-8990, CODEN (USA): CRJHA5.

[14] Ranade,V V (2002), Computational Flow Modelling for Chemical reactor engineering, Academic press.London.

[15] Rong, L et al (2016), Summary of Best Guidelines and Validation of CFD Modeling in Livestock Building to Ensure the Prediction Quality, Computer and Electronics in Agriculture, Vol. 121,No. C, pp. 180-190, DOI: 10.1016/j.compag.2015.12.005.

[16] Shen, F et al (2013), Improvement the Mixing Performance of Rice Straw Anaerobic Digestion for Higher Biogas Production by Computational Fluid Dynamics (CFD) Simulation, Applied Biochemistry and Biotechnology, Vol. 171, No. 3, pp. 626-642, DOI: 10.1007/s12010-013,0375-z.

[17] Terashima, M et al (2008), CFD Simulation of Mixing Anaerobic Digester, Bioresource Technology, Vol. 100, No. 7, pp. 2228-2233, DOI: 10.1016/j.biortech.2008.07.069.

[18] Vavilin V A and Anfelidaki I (2005), Anaerobic Degradation of Solid Material: Importance of Initiation Centres of Methanogenesis, Mixing Intensity and 2D Distribution Model, Biotechnol Bioeng, Vol. 89, No.2, pp. 113-122, DOI: 10.1002/bit.20323.

[19] Vavilin V (2010), Anaerobic Degradation of Organic Waste: An Experience in Mathematical Modeling, Microbiology, Vol. 79, No.3, pp. 334-341, DOI: 10.1134/S0026261710030094.

[20] Vindis, P et al (2009), The Impact of Mesophilic and Thermophilic Anaerobic Digestion on Biogas Production, Journal of Achievements in Materials and Manufacturing Engineering, Vol. 36, No. 2, pp. 192-198.

[21] Wu B and Chen S (2008), CFD Simulation of Non-Newtonian Fluid Flow in Anaerobic Digesters, Biotechnology and Bioengineering, Vol. 99, No.3, pp. 700-711, DOI: 10.1002/bit.21613.

[22] Zhao, X et al (2019), Modified Anaerobic Digestion Model No-1 for Modeling Methane Production from Food Waste in Batch and Semi