www.czasopisma.pan.pl www.journals.pan.pl

JOURNAL OF WATER AND LAND DEVELOPMENT

e-ISSN 2083-4535

Polish Academy of Sciences (PAN) Institute of Technology and Life Sciences – National Research Institute (ITP – PIB)

JOURNAL OF WATER AND LAND DEVELOPMENT
DOI: 10.24425/jwld.2022.143715
2022, Special Issue: 1–7

Assessment of polydisperse substrate flow in a fermentor for computational fluid dynamics modelling

Grzegorz Wałowski

Institute of Technology and Life Sciences – National Research Institute, Falenty, al. Hrabska 3, 05-090 Raszyn, Poland

RECEIVED 05.05.2022 ACCEPTED 02.08.2022 AVAILABLE ONLINE 31.12.2022

Abstract: In this paper study results of selected production methods for agricultural biogas are shown and technical and technological aspects of these methods are described for monosubstrate bioreactors. Based on the available literature, modelling of mixing in bioreactors using computational fluid dynamics (CFD) was is demonstrated. As part of the research, the numerical simulation method was used with a tool that contains CFD codes. The model k-ε is used to simulate the mean flow characteristics under turbulent flow conditions. This is a two-equation model that gives a general description of turbulence. The work presents the results of numerical studies that make it possible to understand the characteristics of fluid flow in the adhesive bed used for the production of agricultural biogas. The tests showed that in the core of the adhesive bed there is a flow of 0.19 m·s^{-1}, while in the outer part of the bed there is a flow in the range 0.01–0.02 m·s^{-1}. Taking into account the substrate inflow of 0.17 m·s^{-1} (in the upper part of the fermentor), it was observed that the Klinkenberg effect for autocyclic movement (from bottom to top) takes place.

The novelty in the article is the observation of the dominant flow in the core of the bed and the autocyclic flow in the opposite direction in the peripheral areas of the adhesive bed.

Keywords: adhesive bed, agricultural biogas, autocyclic flow, bioreactor, computational fluid dynamics (CFD), Klinkenberg phenomenon

INTRODUCTION

Reducing greenhouse gas emissions, increasing renewable energy and improving energy efficiency are the three main goals of future bioenergy systems. Among the available bioenergy technologies, biogas production is also widely used for the management of industrial and municipal organic waste. In the context of biogas production, proper mixing of the organic matter is essential to ensure high biogas yield by the microorganisms. The energy balance of the biogas plant has shown that the power requirement of the agitator accounts for a large part of the total energy consumption of the biogas plant. Testing fermentation chambers to optimise mixing is costly and time consuming. Therefore, computational simulations offer a promising alternative to analyse and improve mixing for performance in agricultural biogas production technology.

Among the various technological methods available [Conti *et al.* 2019] increasing biogas efficiency production by anaerobic digestion (AD), optimisation of the mixing system is one of the most promising approaches [Lebranchu *et al.* 2017; Voytovych *et al.* 2020]. Cost-benefit analyses show that mixing is the largest contributor to the total energy balance of a biogas plant [Naegele *et al.* 2012; Singh *et al.* 2019; Sonnleitner 2012]. Laboratory-scale experiments and computational simulations are a practical and suitable approach for liquid-dynamic mixing tests [Conti *et al.* 2018; Wiedemann *et al.* 2017]. Computational fluid dynamics (CFD) enables detailed modelling of mixing processes under anaerobic conditions [Leonzio 2018]. The state of the art on the use of CFD for bioreactor testing is presented in works [Bridgeman 2012; Dapelo *et al.* 2015; Ding *et al.* 2010; Keshtkar *et al.* 2003; Vesvikar, Al-Dahhan 2005]. To ensure reliable model generation, correct specification of the fermenter, mixer, and initial and boundary conditions is required, and experimental validation is necessary [Lopez-Jimenez *et al.* 2015].

Conti *et al.* [2019] presents a computational fluid dynamics (CFD) model that is used in commercial mixing systems for two propeller agitators arranged in opposite positions of diameter in a container approximately filled 1400 m^3. For the simulation, the rheology of the liquid is adapted to a biomass of 12% by weight. The developed simulation method takes into account the angle of rotation and the height of each propeller. Investigations show that the position of the rotors is away from the lower and large angles of rotation result in favourable fluid dynamics.

One of the most influencing of the AD efficiency is the agitation process as it allows interfacial contact, reduces foam generation and avoids thermal stratification [Meroney, Colorado 2008]. It is worth noting that the suspension of solids in the fermented sludge causes very sticky currents, mixing becomes the basic parameter [Yu *et al.* 2011].

The mixing capacity can also be obtained by increasing the dry matter intake, which reduces operating costs and also increases the quality of the biogas [Wu 2012].

Traditionally design and evaluate different mixing strategies in a fermentor has indicated that experimental and empirical studies have been performed that are not strictly based on physical systems. This method has two major drawbacks [Gallo Molina 2015]:

1) the application of these techniques often leads to suboptimal designs, ultimately increasing equipment costs;
2) experimenting is a costly and time consuming process.

In addition, the fermentation chamber already in operation often has to be switched off for experimentation and optimisation.

CFDs generate results in the right time to experiment and at a lower cost. CFD uses physical equations and the simulation results are more accurate, leading to a better and more detailed understanding of the systems under test. This enables better mixing techniques to be designed and is an effective way to test established mixing strategies.The main objective of work was to evaluate the biogas mixing performance in the fermentation chamber and to investigate the influence of the bubble size on the mixing performance. The relationship between mixture and biogas yield was evaluated with the implementation of the anaerobic digester model (ADM) implementation [Gallo Molina 2015].

CFD is a computational technique that can describe the behaviour of one or more fluids under different conditions using the numerical resolution of a physical equation [Gallo Molina 2015]. CFD modelling is based on the discretisation of the geometric domain of the simulated system – usually by the finite volume method. Then the numerical methods iteratively solve the governing equations. Thus, the exact approximation depends, inter alia, on from: computational time, assumption, boundary conditions and method of equation coupling [Blazek 2001]. The CFD simulation requires at least a numerical solution of the Navier–Stokes equations with pressure coupling and equation of state that closes the degrees of freedom. Please note that the methods used for CFD results are always approximate values. For this reason, it is very important to use the correct equations and in most cases to calibrate the results with the experimental data to ensure the accuracy of the results [Blazek 2001].

AD produces biogas in which sensitive microorganisms are involved. Liquid fermentation must ensure a good heat and mass exchange. To create these fermentation conditions, it is very important to mix Hongguang and Rui [2019] – to simulate CFD the flow field in a fully agitated anaerobic mixing reactor – it is divided into a single layer, open six-blade turbine with baffles and two-layer four-blade 45 degree blades. The results show that an upwardly directed double-layer diagonal bucket may result in a better axial velocity distribution, which favours the formation of a large fluid loop structure that circulates up and down. The average mixing speed of the double layer at 125–320 rpm is lower than the average speed of a single layer, but the mixing speed of the two layers at 60 rpm is higher than that of the single layer.

With the development and wide application of CFD numerical simulation technology, CFD computing technology transfer was used as an effective tool for reaction analysis, monitoring and optimisation of fermentation vessels [Wu 2010]. CFD is based on a numerical solution of equations expressing mass, impulse and energy [Parvareh *et al.* 2010]. Finally, these equations are combined with the fluid transport equation under certain operating conditions to describe the process of mass or heat transfer of a liquid [Karteris *et al.* 2005]. Scientists have done a lot of research on a reactor for producing hydrogen and methane, including heat transfer, mass transfer, mixing, etc. [Vesvikar, Al-Dahhan 2005; Wu 2013]. The flow field in the test center was quantitatively simulated using CFD technology. The ballast reactor, which is equipped with a 6-degree turbine stirrer and two partitions, was used to extract the liquid state data in the reactor. To achieve predictability and controllability, CFD was applied to the design of the AD tank mixing [Hongguang, Rui 2019].

Software supporting the design and optimisation of bioreactors and bioprocesses is used in the field of state-of-the-art computing resources. D'Bastiani *et al.* [2020] used Fluent 16.2 software and this model with particle image velocimetry (PIV) and shadowing techniques. The Euler model, a laminar, three-dimensional model, was used and indefinite simulations were carried out. The results for the mass imbalance of the gaseous and liquid phases, the volume portion of the gas, the gas velocity, the bubble size, the liquid size and the upflow velocity as well as the velocity profiles for the liquid phase were successfully validated on the basis of experimental data. In the dispersed phase, a difference of 4.37% in the volume portion of the gas between the experiments and the simulations was observed. The simulated results showed a difference in the average bubble velocity of 1.73% compared to the shadow images. For the liquid phase, there was a difference of 3.2% in the mean velocity between the simulated and the PIV results. Simulated and experimental velocity profiles showed a better susceptibility inside the reactor. Due to the good agreement between simulations and experiments, the model was considered validated.

The outflow of the gas velocity is closely related to the uplift force, that changes with the cubic power of the bubble diameter – taking this into account, it can be said that the larger the diameter of the bubble, the faster the upward flow velocity of the bubble [Narnoli, Mehrotra 1997]. The experimentally tested gas-liquid-solid fluidisation in light – the velocity of the individual bubbles floating in the suspension – showed a proportional relationship between the size of the bubble and the velocity of the gas flow upwards. [Pourtousi *et al.* 2015; Tsuchiya *et al.* 1997]. The authors investigated the influence of bladder diameter size on the prediction of reference flow using a CFD simulation of a homogeneous follicle column regime [D'Bastiani *et al.* 2020].

Fermenters are often used when the raw material (biomass) to be has a high solids content or a high viscosity. Studies carried out (Chandran *et al.* [2017]) evaluation of the flow characteristics of the vertical fermentation chamber by means of CFD simulation. The simulation results showed that the dissolved content is under normal working conditions and is largely mixed vertically. However, there are many problems with increasing experimental anaerobicity for plant fermentation plants at the field level. One such problem relates to the anaerobic fermentor being agitated, which is necessary to ensure sufficient contact of the bacteria with the substrate in the fermenter. Such situations are well suited to the CFD analysis. Reliable mathematical models are available for these questions.

Knowledge of the rheological properties of the substrate to be processed is still limited. There are examples of the use of fermentation chambers, but so far no CFD simulation study on the effects of biogas recirculation in the fermentation chamber has been published.

The aim of this work was to better understand and improve the mixing process with biogas recirculation in order to improve efficiency future bioreactors. Developed calculation model for the simulation of complex currents in the fermentation chamber, and the CFD simulations on a laboratory scale are discussed. The biogas recirculation properties were also assessed, which provides the knowledge necessary to develop a precise simulation of mixing conditions using an adhesive bed in the reactor.

CFD has become a popular tool for reactor analysis [Latha *et al.* 2009], because it allows to study local conditions in vessels of any size, geometry and operating conditions [Ranade 2002]. The ability of CFD tools to predict mixing behaviour in terms of mixing time, energy consumption, flow patterns and velocity profiles is considered successful at achieving these method objectives, and acceptable results have been achieved in many applications [Wu, Chen 2008]. A literature review of numerical studies of transient gas-liquid flows shows that barbotage columns were simulated using the Euler-Euler approach. In addition, it was observed that the Euler–Lagrange approach is able to predict the average of the properties over time [Lane *et al.* 2002]. This approach allows a simple consideration of the bubble size distribution, which allows a more precise description of the interfacial forces, but with increased computing effort [Buwa 2006; Buwa, Ranade 2002]. The three-dimensional model of the anaerobic reactor represents a typical gas mixing reactor in the laboratory used to generate gaseous hydrogen from municipal solid and liquid waste [Latha *et al.* 2009].

Mixing in AD chambers is necessary to combine bacteria in biomass and food sources in the sediment to stabilise the sediment [Zickefoose, Hayes 1976]. However, the impact of blending on biogas production is not clear – more and more literature suggests that low blending is beneficial [Kaparaju *et al.* 2008; Stroot *et al.* 2001]. This is thought to be the case because high turbulence at high mixing speeds is harmful to methane-producing bacteria [Hoffman *et al.* 2008].

Sindall *et al.* [2013] used CFDs to model a mechanical mixing fermenter in the laboratory and to investigate turbulence-kinetic energy patterns at different mixing speeds. The results of the four turbulence models are compared with the PIV results and the achievable k-ε (k = turbulent kinetic energy, ε = rate of dissipation of turbulent kinetic energy)value seems to be the best for predicting flow patterns in the fermentation chamber. The work uses CFD simulations of a mechanically mixed fermentation chamber on a laboratory scale to identify areas of high and low turbulence in mixed fermentation chambers with different speeds. This will allow a better understanding of the relationship between blending and biogas production.

Zhang *et al.* [2016] compared the flow field and impeller power consumption with different feed materials, using CFD [Dabiri *et al.* 2021]. They used the k-ε standard turbulence model simulating the tank mixed by the rotor and verified the numerical results with the experimental data. The ability of CFD to model mixing in bioreactors by means of an impeller to mix the contents of the fermentation tank, showed the non-Newtonian behaviour of the raw material and confirmed the results by comparing the numerical value of the power consumption of the mixer derived and the energy dissipation rate function with the value obtained on the basis of experimental data [Bridgeman 2012]. For mixing the raw material inside the digestion chamber, gas injection as well as fluid recirculation systems are used in some cases [Saini *et al.* 2020]. Euler–Lagrange's innovative CFD approach to fluid flow simulation was noted in a fermentation tank that uses gas injection as a mixer [Dapelo *et al.* 2015]. In this way, the accuracy of the results was assessed by visualising the flow field in a laboratory model with PIV measurement. A fermentation reactor model was developed [Sajjadi *et al.* 2016], which uses fluid injectors to recirculate the biomass. Thus, the importance of the location of both inlet and outlet fluid streams was demonstrated. López-Jiménez *et al.* [2015] simulated a mixing process in a fermentation tank where the sludge was returned to the tank at high speeds. They modelled pump inlets with different entry angles and nozzle shapes to accelerate the inlet velocity. Reynolds averaged Navier–Stokes (RANS) equations were solved in a single-phase CFD model taking into account both Newtonian and non-Newtonian sludge properties. Tests of Meroney and Sheker [2014], Hernandez-Aguilar *et al.* [2016], Low *et al.* [2017] and Meister *et al.* [2018] agitated tanks have concentrated on rotary mixers located at the center of the tank, but the mixing quality (i.e. dead volume, mix time, speed gradient etc.) of asymmetric mixers is largely unknown.

Therefore, in development of Dabiri *et al.* [2021] the mixing quality is carefully checked with a mixer placed asymmetrically on one side of the tank. Moreover, since the energy consumption of the mixer has a significant impact on the overall energy efficiency of digesters, there is a need to gain an insight into the energy consumption of the mixer and its relation to mixing performance. The test is aimed at assessing the energy consumption of the mixer, in addition to the dead volume and the speed gradient, ensuring the mixing efficiency. Other important factors are the non-Newtonian characteristics and the rotational speed of the stirrer, the influence of which on the mixing efficiency is investigated. After designing the model geometry, including the effective components for mixing quality, the mesh was implemented. Then, to determine the amount of dead volume and the ratio of the stirrer speed and energy consumption, the velocity and pressure fields are obtained by solving the fluid flow equation, based on the SIMPLE algorithm, taking into account the non-Newtonian characteristics of the fluid.

The experimental studies concerned the evaluation of the hydrodynamics of the polydisperse substrate flow in the bioreactor for the CFD model.

Numerical tests were carried out in the field of:

- evaluation of the geometry of the analysed fermentor,
- isosurface analysis for flow velocity,
- gas permeability assessment of the adhesive bed.

MATERIALS AND METHODS

Biogas production in real conditions – described in detail in the paper by Klimek *et al.* [2021] – located on a farm dealing with the breeding of fattening pigs [DANBRED 2021; Wałowski 2021b]. The geometry of the object was developed for the functioning installation [Wałowski *et al.* 2019], shown in Figure 1 – the fermenter (bioreactor) was created in the program Space Claim Direct Modeller Ansys Fluent R 19.2 on the basis of the working documentation of the fermenter [Wałowski 2019].

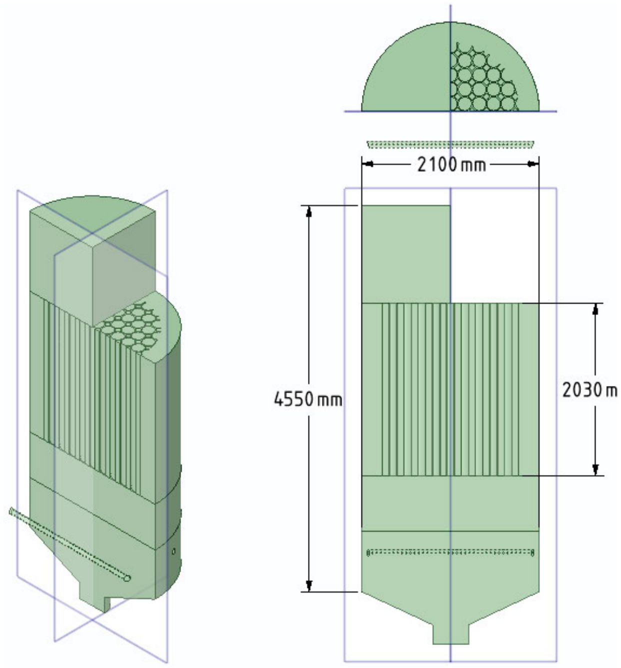

Fig. 1. Geometry of the analysed fermentor: upper part – storage area of the produced biogas; middle part – adhesive bed forms a skeleton composed of 72 pipes; lower part – fermenter drain; source: own elaboration

The geometry includes the inlet and outlet tube of the substrate (suspension), the tube bundle and the air bubble tubes – Figure 1. The numeric grid was created in the Ansys Fluent R 19.2 The geometry is divided into smaller, simpler blocks – hexagonal mesh for most volumes. The flow model was defined in the commercial fluid simulation software Ansys Fluent R 19.2. Basic information about the flow model:

a) turbulent flow – the Reynolds averaged Navier–Stokes model (RANS) with enhanced wall processing option;
b) two-phase liquid volume approach (VOF) with clear wording;
c) SIMPLE calculation algorithm (semi-implicit method for pressure-dependent equations);
d) discretisation of the second series of the impulse equation;
e) a georeconstructed algorithm for tracing border crossings;
f) a career controlled by residual observations.

Liquid parameters (based on the results of own measurements):

a) pig manure: density 998.2 kg·m^{-3}, viscosity 0.0015 Pa·s (50% higher than water);
b) gas modelled as air: density 1.225 kg·m^{-3}, viscosity 0.018 mPa·s;
c) external forces: gravity 9.81 m·s^{-2}.

Boundary conditions:

a) manure intake: 7.209 kg·s^{-1};
b) turbulence intensity at the input: 5%;
c) turbulent viscosity ratio at inlet: 10.

Mixing took place cyclically every 4 h for 10 min, while the drain (filling) in the amount of 1500 dm^3 took place every 7 days.

RESULTS AND DISCUSSION

The results were analysed and the following visualisations were created. The area where the suspension current enters the fermenter is shown in Figure 2a. You can see that the main flow takes place in some selected pipes that make up the adhesive bed. The flow lines are shown in Figure 2b, mixing can be observed in

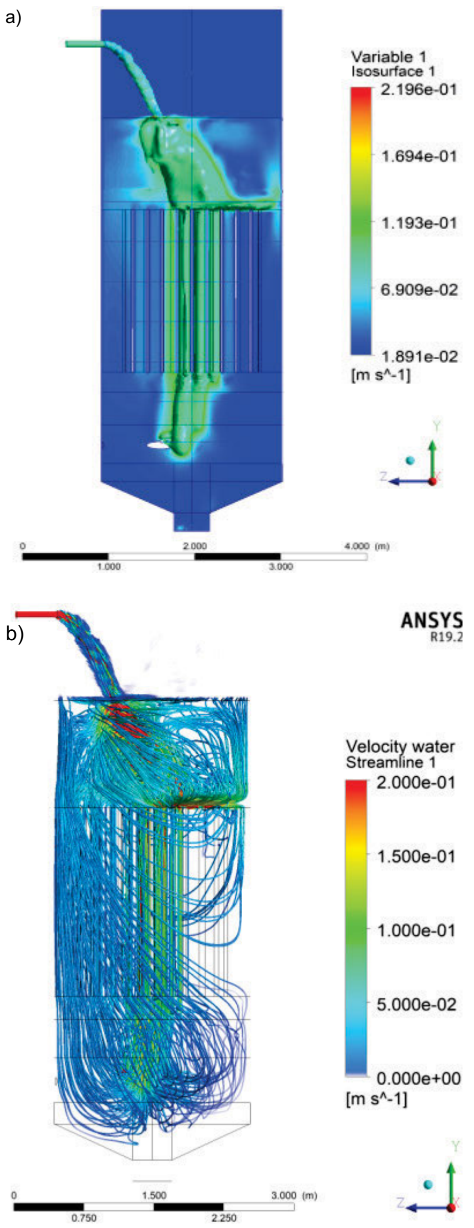

Fig. 2. Numerical evaluation of the flow in the bioreactor: a) velocity iso-surface: qualitative representation of the main mixing range and the location where the highest speeds occur; b) streamlines – view of the recirculation zone; source: own study

most of the column volume, with the exception of the area at the bottom of the column facing the inlet pipe. A recirculation zone was also observed, indicating a very limited mixing in this part of the fermenter (bioreactor).

It should be noted that the dominant flow in the bed takes place in the central part of the adhesive bed (bed core) and takes place from top to bottom – Figure 3. On the other hand, for the remainder of the bed, the flow is from bottom to top – thus creating an autocyclic movement characteristic of the Klinkenberg phenomenon [Klinkenberg 1941; Wałowski, Filipczak 2017]. Since the adhesive bed intensively contributes to the foaming of the polydispersive substrate, as a result of which the skeleton bed with porous walls shows effective gas permeability resulting from the sliding of gas molecules (methane) on the walls of porous channels.

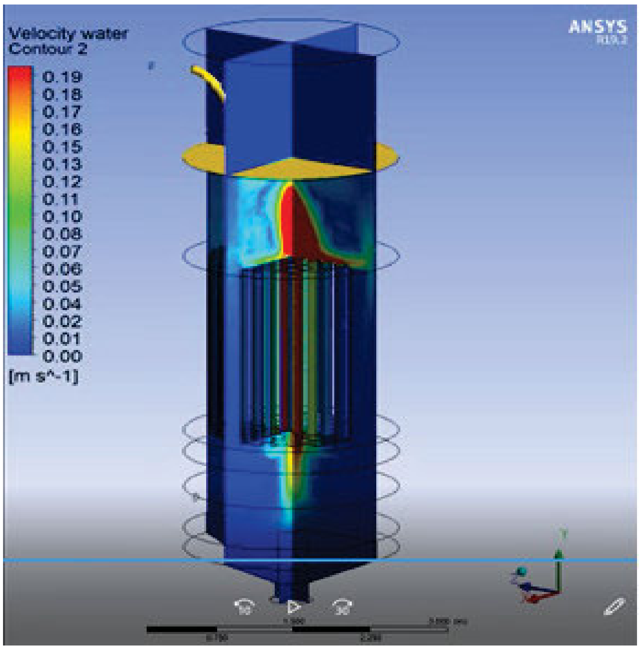

Fig. 3. Numerical evaluation of the flow in the bioreactor – the dominant flow in the bed core and the autocyclic flow in the opposite direction in the peripheral areas of the adhesive bed; source: own study

The analysis of the results shows the limitations of the simulation that affect the accuracy:

- the suspension is modelled as a liquid of constant viscosity (properties of a Newtonian liquid);
- small gaps between the pipes were omitted when modelling the tube bundle.

The polydisperse substrate has the greatest impact on the accuracy of modelling, and the obtained results provide qualitative information about the process of hydrodynamic mixing in the skeletal fermenter [Wałowski 2021a].

CONCLUSIONS

The introduction of agricultural biogas plants requires further research into the definition of basic indicators for determining the energy intensity of different technologies, comparing and selecting them in terms of environmental, economic and energy costs.

In the context of hydrodynamic conditions, the flow of polydisperse substrate through the adhesive bed was assessed using the CFD method. It was found that the cyclic mixing system plays a decisive role in the production of biogas with the use of an adhesive bed. Considering the use of computational fluid dynamics, a circulation zone was observed, indicating a very limited mixing in this part of the fermenter. It should be noted that the dominant flow in the bed takes place in the central part of the adhesive bed (bed core) and takes place from top to bottom. On the other hand, for the remainder of the bed, the flow is from bottom to top – thus creating an autocyclic movement characteristic of the Klinkenberg phenomenon.

ACKNOWLEDGMENTS

The study carried out in the context of a the National Centre for Research and Development realised in the BIOSTRATEG programme, contract No. BIOSTRATEG1/269056/5/NCBR/2015 with 11.08.2015.

REFERENCES

Blazek J. 2001. Computational fluid dynamics: Principles and applications. Amsterdam. Elsevier. ISBN 978-0-08-044506-9 pp. 470. DOI 10.1016/B978-0-08-044506-9.X5000-0.

Bridgeman J. 2012. Computational fluid dynamics modeling of sewage sludge moxing in an anaerobic digester. Advances in Engineering Software. Vol. 44(1) p. 54–62. DOI 10.1016/j.advengsoft.2011.05.037.

Buwa V.V. 2006. Eulerian-Lagrangian simulations of unsteady gas-liquid flows in bubble columns. International Journal of Multiphase Flow. Vol. 32 p. 864–885. DOI 10.1016/j.ijmultiphaseflow.2006.02.017.

Buwa V.V., Ranade V.V. 2002. Dynamics of gas-liquid flow in a rectangular bubble column: Experiments and single/multi-group CFD simulations. Chemical Engineering Science. Vol. 57 (22–23) p. 4715–4736. DOI 10.1016/S0009-2509(02)00274-9.

Chandran J., Yogaraj D., Manikandan K., Jeyaraman P. 2017. Optimization of bio gas recirculation velocity in biogas mixing anaerobic digester with the feed of 8% Tds using CFD [online]. International Journal of Mechanical Engineering and Technology. Vol. 8(8) p. 596–606. [Access 27.04.2022]. Availble at: https://iaeme.com/MasterAdmin/Journal_uploads/IJMET/VOLUME_8_ISSUE_8/IJMET_08_08_065.pdf

Conti F., Saidi A., Goldbrunner M. 2019. CFD Modelling of biomass mixing in anaerobic digesters of biogas plants. Environmental and Climate Technologies. Vol. 23(3) p. 57–69. DOI 10.2478/rtuect-2019-0079.

Conti F., Wiedemannn L., Saidi A., Sonnleitner M., Goldbrunner M. 2018. Mixing of a model substrate in a scale-down laboratory digester and processing with a computational fluid dynamics model. Procedigs. of 26th EUBCE – European Biomass Conference and Exhibition. Copenhagen 2018 p. 811–815. DOI 10.5071/26thEUBCE2018-2CV.5.34.

D'Bastiani E., Campião K., Boeger W., Araújo S. 2020. The role of ecological opportunity in shaping host–parasite networks. Parasitology. Vol. 147(13) p. 1452–1460. DOI 10.1017/S003118202000133X.

Dabiri S., Noorpoor A., Arfaee M., Kumar P., Rauch W. 2021. CFD Modeling of a stirred anaerobic digestion tank for evaluating energy consumption through mixing. Water. Vol. 13, 1629. DOI 10.3390/w13121629.

DANBRED 2021. Danbred genes for global pig production [online]. [Access 20.05.2021]. Available at: https://danbred.com/our-many-years-of-breeding-progress-contributes-to-your-future-pig-production/

Dapelo D., Alberini F., Bridgeman J. 2015. Euler-Lagrange CFD modelling of unconfined gas mixing in anaerobic digestion. Water Research. Vol. 85 p. 497–511. DOI 10.1016/j.watres.2015.08.042.

Ding J., Wang X., Zhou X.-F., Ren N.-Q., Guo W.-Q. 2010. CFD optimization of continuous stirred-tank (CSTR) reactor for biohydrogen production. Bioresource Technology. Vol. 101 p. 7005–7013. DOI 10.1016/j.biortech.2010.03.146.

Gallo Molina J.P. 2015. CFD modelling of biogas mixing in an anaerobic digester used in wastewater treatment. Bogotá. Universidad de los Andes pp. 37.

Hernandez-Aguilar E., Alvarado-Lassman A., Osorio-Mirón A., Méndez-Contreras J.M. 2016. Development of energy efficient mixing strategies in egg-shaped anaerobic reactors through 3D CFD simulation. Journal of Environmental Science and Health. Vol. 51(7) p. 536–543. DOI 10.1080/10934529.2016.1141619.

Hoffman R., Garcia M.L., Vesvikar M., Karim K., Al-Dahhan M.H., Angenent L.T. 2008. Effect of shear on performance and microbial ecology of continuously stirred anaerobic digesters treating animal manure. Biotechnology and Bioengineering. Vol. 100 p. 38–48. DOI 10.1002/bit.21730.

Hongguang Z., Rui J. 2019. CFD simulation study on mixing experiment of anaerobic digestion tank. E3S Web of Conferences. Vol. 118, 02047 p. 1–7. DOI 10.1051/e3sconf/201911802047.

Kaparaju P., Buendia I., Ellegaard L., Angelidaki I. 2008. Effects of mixing on methane production during thermophilic anaerobic digestion of manure: Lab-scale and pilotscale studies. Bioresource Technology. Vol. 99 p. 4919–4928. DOI 10.1016/j.biortech.2007.09.015.

Karteris A., Papadopoulos A., Balafoutas G. 2005. Modeling the temperature pattern of a covered anaerobic pond with computational fluid dynamics. Water, Air and Soil Pollution. Vol. 162(1–4) p. 107–125. DOI 10.1007/s11270-005-5996-6.

Keshtkar A., Meyssami B., Abolhamd G., Ghaforian H., Khalagi Asadi M. 2003. Mathematical modeling of nonideal mixing continuous flow reactors for anaerobic digestion of cattle manure. Bioresource Technology. Vol. 87(1) p. 113–124. DOI 10.1016/S0960-8524(02)00104-9.

Klimek K., Kapłan M., Syrotyuk S., Konieczny R., Anders D., Dybek B., Karwacka A., Wałowski G. 2021. Production of agricultural biogas with the use of a hydrodynamic mixing system of a polydisperse substrate in a reactor with an adhesive bed. Energies. Vol. 14, 3538. DOI 10.3390/en14123538.

Klinkenberg L.J. 1941. The permeability of porous media to liquids and gases [online]. In: API drilling and production practice p. 200–213. [Access 10.04.2022]. Available at: https://faculty.ksu.edu.sa/sites/default/files/klinkenbergspaper-1941.pdf

Latha S., Borman D.J., Sleigh P.A. 2009. CFD multiphase modelling for evaluation of gas mixing in an anaerobic digester [online]. In: 14th European Biosolids and Organic Resources Conference and Exhibition. 9–11.11.2009. Leeds, UK. The Royal Armouries p. 1–16. [Access 20.05.2022]. Available at: https://eprints.whiterose.ac.uk/10314/1/AD_Shanmugham_L_BIOSOLIDS_09.pdf

Lane G.L., Schwarz M.P., Evans G.M. 2002. Predicting gas-liquid flow in a mechanically stirred tank. Applied Mathematical Modelling. Vol. 26 p. 223–235. DOI 10.1016/S0307-904X(01)00057-9.

Lebranchu A., Delaunay S., Marchal P., Blanchard F., Pacaud S., Fick M., Olmos E. 2017. Impact of shear stress and impeller design on the production of biogas in anaerobic digesters. Bioresource Technology. Vol. 245(A) p. 1139–1147. DOI 10.1016/j.biortech.2017.07.113.

Leonzio G. 2018. Study of mixing systems and geometric configurations for anaerobic digesters using CFD analysis. Renewable Energy. Vol. 123 p. 578–589. DOI 10.1016/j.renene.2018.02.071.

Lopez-Jimenez P.A., Escudero-González J., Martínez T.M., Montañana V.F., Gualtieri C. 2015. Application of CFD methods to an anaerobic digester: The case of Ontinyent WWPT, Valencia, Spain. Journal of Water Process Engineering. Vol. 7 p. 131–140. DOI 10.1016/j.jwpe.2015.05.006.

Low S.C., Eshtiaghi N., Shu L., Parthasarathy R. 2017. Flow patterns in the mixing of sludge simulant with jet recirculation system. Process Safety and Environmental Protection. Vol. 112 p. 209–221. DOI 10.1016/j.psep.2017.08.016.

Meister M., Rezavand M., Ebner C., Pümpel T., Rauch W. 2018. Mixing non-Newtonian flows in anaerobic digesters by impellers and pumped recirculation. Advances in Engineering Software. Vol. 115 p. 194–203. DOI 10.1016/j.advengsoft.2017.09.015.

Meroney R., Colorado P. 2008. CFD simulation of mechanical draft tube mixing in anaerobic digester tanks. Water Research. Vol. 43 (4) p. 1040–1050. DOI 10.1016/j.watres.2008.11.035

Meroney R.N., Sheker R.E. 2014. CFD simulation of vertical linear motion mixing in anaerobic digester tanks. Water Environment Research. Vol. 86 p. 816–827. DOI 10.2175/106143014x14062131177836.

Naegele H.J., Lemmer A., Oechsner H., Jungbluth T. 2012. Electric energy consumption of the full scale research biogas plant ‘unterer Lindenhof’: Results of longterm and full detail measurements. Energies. Vol. 5(12) p. 5198–5214. DOI 10.3390/en5125198.

Narnoli S.K., Mehrotra I. 1997. Sludge blanket of UASB reactor: Mathematical simulation. Water Research. Vol 31(4) p. 715–726. DOI 10.1016/S0043-1354(97)80987-6.

Parvareh A., Rahimi M., Alizadehdakhel A., Alsairafi A.A. 2010. CFD and ERT investigations on two-phase flow regimes in vertical and horizontal tubes. International Communications in Heat and Mass Transfer. Vol. 37(3) p. 304–311. DOI 10.1016/j.icheatmasstransfer.2009.11.001.

Pourtousi M., Ganesan P., Sahu J.N. 2015. Effect of bubble diameter size on prediction of flow pattern in Euler–Euler simulation of homogeneous bubble column regime. Measurement. Vol. 76 p. 255–270. DOI 10.1016/j.measurement.2015.08.018.

Ranade V.V. 2002. Computational flow modelling for chemical reactor engineering [eBook]. London. Academic Press. ISBN 9780080502298 pp. 401.

Saini A.K., Paritosh K., Singh A.K., Vivekanand V. 2020. CFD approach for pumped-recirculation mixing strategy in wastewater treatment: Minimizing power consumption, enhancing resource recovery in commercial anaerobic digester. Journal of Water Process Engineering. Vol. 40, 101777. DOI 10.1016/j.jwpe.2020.101777.

Sajjadi B., Raman A.A.A., Parthasarathy R. 2016. Fluid dynamic analysis of non-Newtonian flow behavior of municipal sludge simulant in anaerobic digesters using submerged, recirculating jets. Chemical Engineering Journal. Vol. 298 p. 259–270. DOI 10.1016/j.cej.2016.03.069.

Sindall R., Bridgeman J., Carliell-Marquet C. 2013. CFD modelling of lab-scale anaerobic digesters to determine experimental sampling locations [online]. 4th Annual BEAR PGR Conference 2013. University of Birmingham, UK p. 1–8. [Access 15.05.2022].

Available at: https://intranet.birmingham.ac.uk/it/teams/infrastructure/research/bear/documents/public/bear-pgr-2013-history/presentations/SindallR.pdf

Singh B., Szamosi Z., Simenfalvi Z. 2019. State of the art on mixing in an anaerobic digester: A review. Renewable Energy. Vol. 141 p. 922–936. DOI 10.1016/j.renene.2019.04.072.

Sonnleitner M. 2012. Ecological and economic optimization of biogas plants. MPhil Thesis. Leicester. De Montfort University pp. 171.

Stroot P.G., McMahon K.D., Mackie R.I., Raskin L. 2001. Anaerobic codigestion of municipal solid waste and biosolids under various mixing conditions – I: digester performance. Water Research. Vol. 35 p. 1804–1816. DOI 10.1016/S0043-1354(00)00439-5.

Tsuchiya K., Furumoto A., Fan L., Zhang J. 1997. Suspension viscosity and bubble rise velocity in liquid-solid fluidized beds. Chemical Engineering Science. Vol. 52(18) p. 3053–3066. DOI 10.1016/S0009-2509(97)00127-9.

Vesvikar M.S., Al-Dahhan M. 2005. Flow pattern visualization in a mimic anaerobic digester using CFD. Biotechnology in Bioengineering. Vol. 89(6) p. 719–732. DOI 10.1002/bit.20388.

Voytovych I., Malovanyy M., Zhuk V., Mukha O. 2020. Facilities and problems of processing organic wastes by family type biogas plants in Ukraine. Journal of Water and Land Development. No. 45 p. 185–189. DOI 10.24425/jwld.2020.133493.

Wałowski G. 2019. Multi-phase flow assessment for the fermentation process in mono-substrate reactor with skeleton bed. Journal of Water and Land Development. No. 42 p. 150–156. DOI 10.2478/jwld-2019-0056.

Wałowski G. 2021a. Assessment of the flow of substrate and agricultural biogas through the adhesive skeleton bed in phenomenological and numerical terms. Archives of Thermodynamics. Vol. 42(3) p. 243–253. DOI 10.24425/ather.2020.138118.

Wałowski G. 2021b. Development of biogas and biorafinery systems in Polish rural communities. Journal of Water and Land Development. No. 49 p. 156–168. DOI 10.24425/jwld.2021.137108.

Wałowski G., Borek K., Romaniuk W., Wardal W.J., Borusewicz A. 2019. Nowoczesne systemy pozyskania energii – biogazu [Modern systems of obtaining energy – Biogas]. Łomża. Wydaw. WSA w Łomży. ISBN 978-83-947669-9-3 pp. 116.

Wałowski G., Filipczak G. 2017. Klinkenberg effect in hydrodynamics of gas flow through anisotropic porous materials. EEMS 2017, E3S Web of Conferences. Vol. 19, 03008. DOI 10.1051/e3sconf/20171903008.

Wiedemann L., Conti F., Sonnleitner T.J.M., Zörner W., Goldbrunner M. 2017. Mixing in biogas digesters and development of an artificial substrate for laboratory-scale mixing optimization. Chemical Engineering & Technology. Vol. 40 p. 238–247. DOI 10.1002/ceat.201600194.

Wu B. 2010. CFD simulation of gas and non-Newtonian fluid two-phase flow in anaerobic digesters. Water Research. Vol. 44(13) p. 3861–3874. DOI 10.1016/j.watres.2010.04.043.

Wu B. 2012. CFD simulation of mixing for high-solids anaerobic digestion. Biotechnology & Bioengineering. Vol. 109(8) p. 2116–2126. DOI 10.1002/bit.24482.

Wu B. 2013. Advances in the CFD to characterize design and optimize bioenergy systems. Computers and Electronics in Agriculture. Vol. 93 p. 195–208. DOI 10.1016/j.compag.2012.05.008.

Wu B., Chen S. 2008. CFD simulation of non-Newtonian fluid flow in anaerobic digesters. Biotechnology and Bioengineering. Vol. 99 (3) p. 700–711. DOI 10.1002/bit.21613.

Yu L., Ma J., Chen S. 2011. Numerical simulation of mechanical mixing in high solid anaerobic digester. Bioresource Technology. Vol. 102 (2) p. 1012–1018. DOI 10.1016/j.biortech.2010.09.079.

Zhang Y., Yu G., Yu L., Siddhu M.A.H., Gao M., Abdeltawab A.A., Al-Deyab S.S., Chen X. 2016. Computational fluid dynamics study on mixing mode and power consumption in anaerobic mono- and co-digestion. Bioresource Technology. Vol. 203 p. 166–172. DOI 10.1016/j.biortech.2015.12.023.

Zickefoose C., Hayes R.B.J. 1976. Anaerobic sludge digestion: Operations manual. USEPA pp. 192.