INTERNATIONAL JOURNAL OF
ENERGY AND ENVIRONMENT

Volume 2, Issue 6, 2011 pp.963-974
Journal homepage: www.IJEE.IEEFoundation.org

Modeling flow inside an anaerobic digester by CFD techniques

Alexandra Martínez Mendoza[1], Tatiana Montoya Martínez[2], Vicente Fajardo Montañana[2], P. Amparo López Jiménez[1]

[1] Departamento de Ingeniería Hidráulica y Medio Ambiente, Universitat Politècnica de València, Camino de Vera S/N 46022, Spain.
[2] Grupo Aguas de Valencia. Avenida Marqués del Turia 19 46005 Valencia, Spain.

Abstract

Anaerobic processes are used to treat high strength organic wastewater as well as for the treatment of primary and secondary sludge from conventional wastewater treatment plants. In these processes, heterotrophic microorganisms convert biodegradable organic matter to methane and carbon dioxide in the absence of dissolved oxygen and nitrate.

Some of the most important aspects of the design of anaerobic digesters are related to hydraulic considerations. In spite of its important role in performance, hydraulics of flow inside digesters has not been quantified or adequately characterized. In this contribution a three-dimensional steady-state computational fluid dynamics (CFD) simulation has been performed for a particular anaerobic digester, in order to visualize the flow patterns. Flow and velocities profiles have been represented inside the digester to identify possible dead zones or stratifications. The geometry of a real digester installed in Valencia Waste Water Treatment Plant (located in Quart-Benager, Valencia, Spain) has been used in order to consider the proposed methodology.

Keywords: Anaerobic digester, Computational fluid dynamics, Hydrodynamic model, Velocities profiles.

1. Introduction

Anaerobic digestion is the degradation of organic matter by microorganisms that do not require oxygen for degradation of such material. Recently anaerobic digestion processes have taken much force, for the excellent results obtained by producing a very stable sludge, but it is still an ongoing study looking for improvements because it is a sensitive system that can be seen disturbed and not properly working. One of the important reasons for failure is improper mixing provided in the digester. Proper mixing is needed for optimal performance and operation since mixing or agitation is required to homogenize the contents of digesters, to ensure uniform distribution of substrate and microorganisms culture, to avoid settling of the heavy solid particles to the bottom, to avoid flotation of biomass at the surface of the sludge, and to maintain the desired pH and temperature of the slurry for the microbial processes [1].

An understanding of the hydrodynamic behavior of anaerobic reactors is considered to be of paramount importance for optimization of the process, enabling the detection and solution of problems that often occur due to reactor design and geometrical aspects. The need for efficient mixing inside anaerobic reactors has been recognized and stressed in the technical literature [2].

Nowadays, models become a more powerful tool for representing these digesters behavior. Recently, references can be found related to digestion procedures optimization by analyzing the adequate mixing in environment of anaerobic bacteria. [3]. Even some modeling studies can be found by using dynamic models to investigate anaerobic digestion performance [2], in these studies, the importance of mixing indicators such as retention time and the degree of mixing have been pointed out.

A 3-D flow model was developed by Fleming [4], considering a wide range of aspects such as bulk fluid motion, sedimentation, bubble mixing, bubble entrainment, and buoyant mixing to simulate the flow patterns of a covered lagoon digester.

The importance of defining performance indicators has been documented by Keshtkar et al. in [5]. They proposed two characteristic mixing parameters: the relative volume of the flow-through region, and the ratio of the internal exchange flow rate to the feed flow rate for evaluating the digestion performance. In this sense, Pena et al. [6] studied modified pilot-scale anaerobic ponds receiving domestic sewage with different configurations and flow rates.

First 3-D steady-state computational fluid dynamics (CFD) simulations were presented in Vesvikar and Al-Dahhan [7] to visualize the flow patterns and to obtain hydrodynamic parameters. Also, in Vesvikar and Al-Dahhan and Vesvikar et al. [8] computer automated radioactive particle tracking and computed tomography were used to visualize the flow patterns and obtain hydrodynamic parameters in mimic anaerobic digesters. Karim et al. [9-12] experimentally studied the effect of mixing (biogas recirculation, impeller, and slurry recirculation) on the anaerobic digestion of animal waste for lab-scale digesters. Binxin Wu [13] implemented a CFD model to predict the fields of flow in a mixing of an anaerobic digester, considering non Newtonian Fluid, and obtaining completely different results.

Therefore, the present work is focused on modeling fluid dynamics within the digester and defining parameters to identify the mixing inside it. The main objective of this work is the modeling of the digester hydrodynamics in order to study and identify the mixing zones and zones of nor or little mixing mixture that we call dead zones. CFD represents the possibility of visualizing the flow inside the digester, and analyze in detail the mixture and parameters that represent this dynamics.

2. Model development

2.1 CFD model for representing dynamics inside the digester

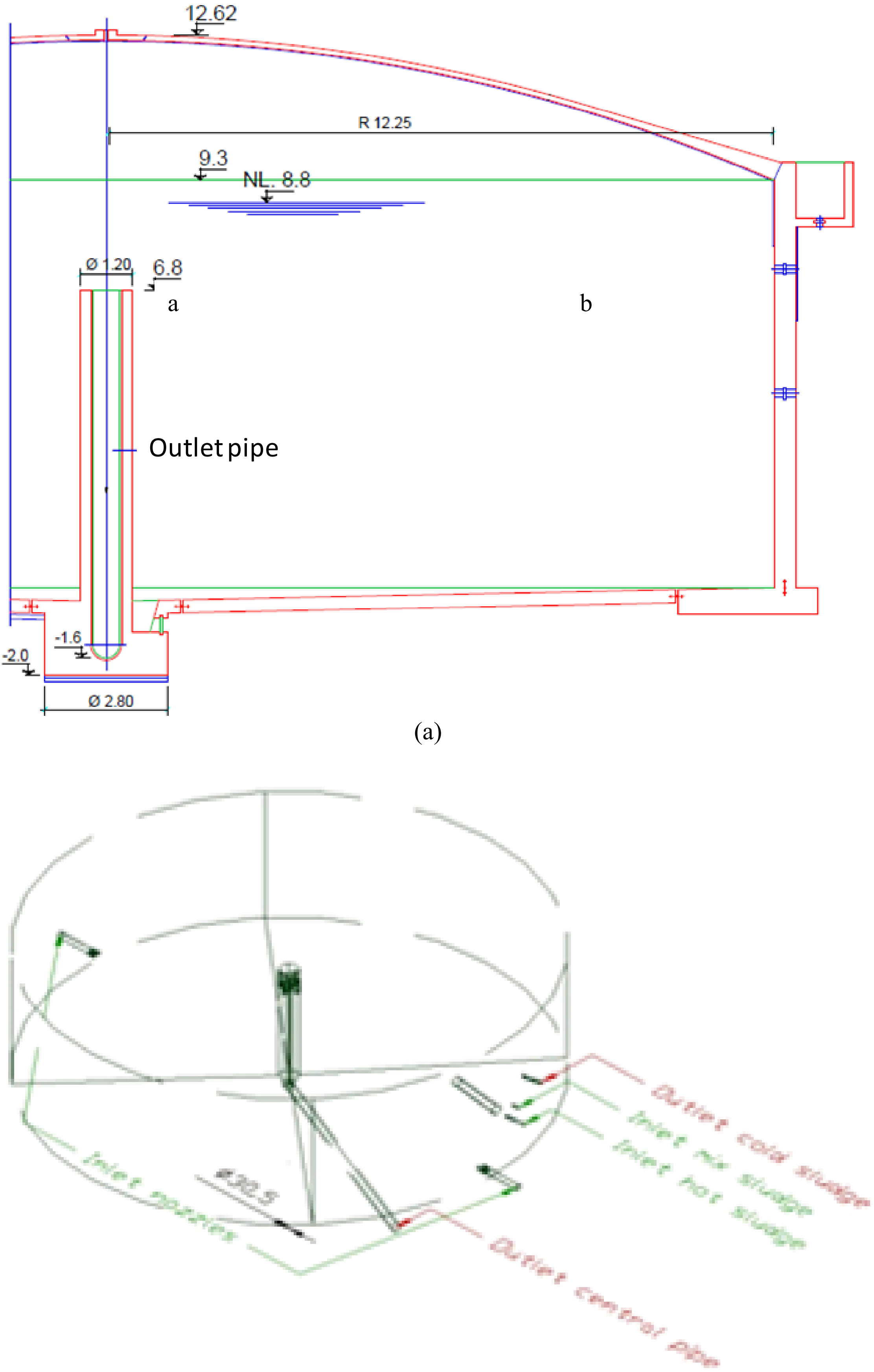
Computational fluid dynamics (CFD) proves to be a valuable and efficient tool to understand and evaluate the fluid dynamics of flow systems. This model has been development using a CFD commercial package: Star CCM+. This model solves numerically the laws governing fluid dynamics, solving equations by a geometric domain. The basic flow equations used are the Navier-Stokes, Viscous Fluxes in laminar flow and turbulent, and momentum equation in discrete form. The magnitudes of velocity, pressure and temperature are calculated in a discrete manner at the nodes of a mesh or network, describing the flow geometry modeling [14].

2.2 Geometry

The model developed is based on a primary digester Quart-Benager WWTPs. The digester dimensions are 30,50m in diameter and 8,80m high cylindrical, with a total volume of 6.504 m^3. It is structurally built of reinforced concrete with walls 0,50m thick. (See Figure 1). The removal of sludge is done by the fund, through a set of valves and there is also a shot at half height for sludge extraction led to the secondary digester.

The agitation system of the sludge is carried out using the DINOMIX system; this is a mixing device by means of a jet, causing a stream into the digester with a drag due to the forces resulting from the viscosity and flow jet velocity. The forces thus come into play, and it creating a speed zone of instability that results in a turbulent area.

With the real dimensions of the digester a replica of the geometry was created using AutoCAD 3D software, and then converted the file type lithography in order to be implemented in STAR CCM+ Software.



(a)

(b)

Figure 1. (a) Dimensions of the geometry of anaerobic digester; (b) Inlet and outlet of anaerobic digester

2.3 Mesh

Mathematical model is created from the geometric model, called meshing or discretization of the continuum, generating a system of equations. Several simplifications were made to develop the model to better connection with CFD: The affluent is performed by 2 nozzles DINOMIX, with same input velocity, and two pipes to feed a digester with mix mud and hot mud. The output of the mud is made by emptying from a central pipe and other lateral out by to extract hot mud. The output for overflow is omitted. Variations of temperature are not considered, only flows with fixed conditions of entry. Geometry of the digester was maintained in all the models, only changing the angles of the nozzles. See Figure 2.

Conditions of meshing are defined: surface remesher, volume polyhedral mesher, and prism layer mesher. The surface mesh has generated 177.412 faces and volume mesh has generated 848.729 cells,. The obtained mesh is very precise with cell size of 0.22 m it was appropriate for the dimensions of model.

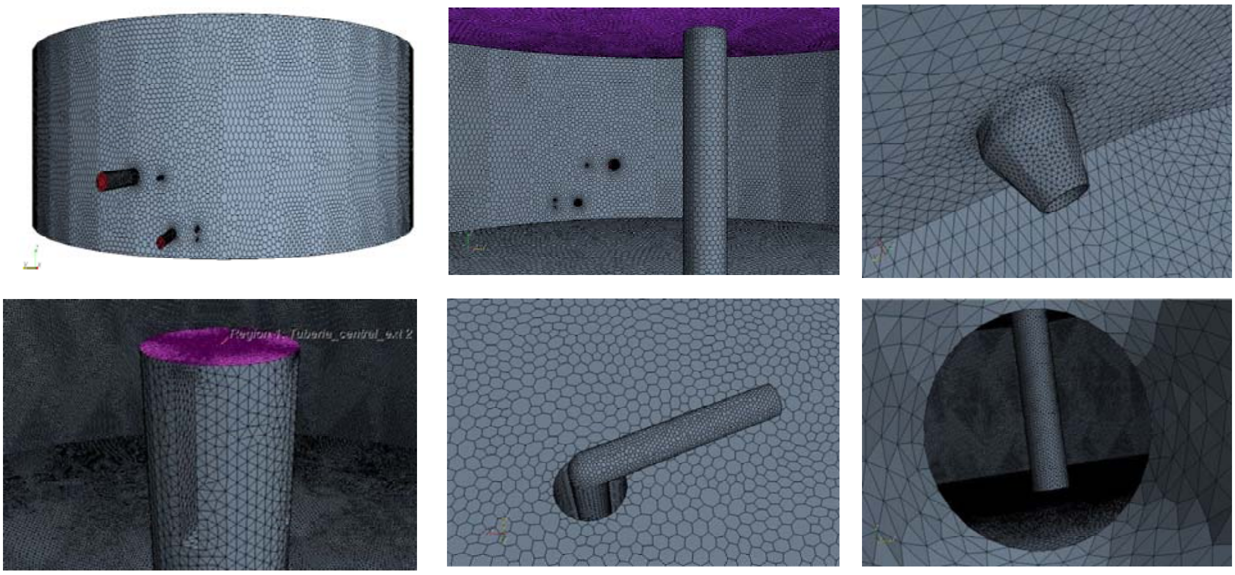


Figure 2. Details on volume mesh

2.3 Boundary conditions

A classical velocity inlet boundary condition has been used. The outlet was first defined as a pressure outlet, the rest of geometry is represented by boundaries conditions of wall. Figure 3 shows the boundary conditions used in the volume of the problem being solved. Physical conditions in the model are namely: 3D model. steady state.

The physical properties of a continuous fluid with density (999.66 kg/m^3) and dynamic viscosity (0.065 Pa-s), sludge coming from Quart-Benager WWTPs. Segregated flow. Constant density. Turbulent flow using the standard k-ε model. This standard model k-ε is a semi-empirical model based on transport equations for kinetic energy turbulent (k) and the range of dissipation (ε) [15].

After model conditions have been defined, several iterations were performed, which results in velocity. Turbulent conditions proposed the better convergence solutions as shown in Figure 4, with a smaller value for final residuals.

3. Results

Once the system has been designed, it has confirmed the efficiency of the mixture to be looking. The final results means values about the hydrodynamic of the sludge inside the digester, as velocity fields, percentage of volume with dead zones and mixing zones. Further the flow pattern is also studied by means of parameters as the geometry influence and turbulence variations.

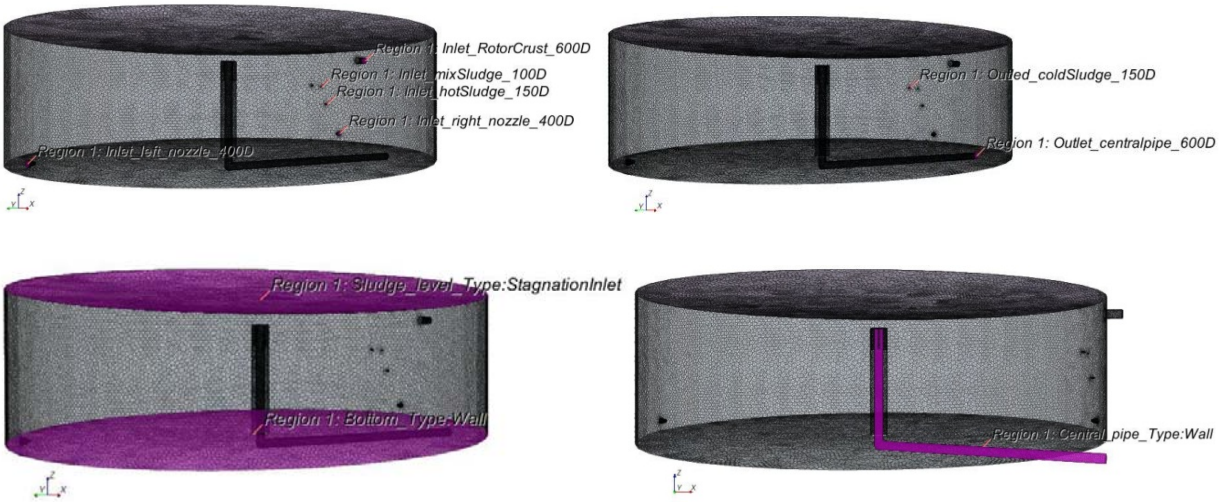


Figure 3. Boundary conditions

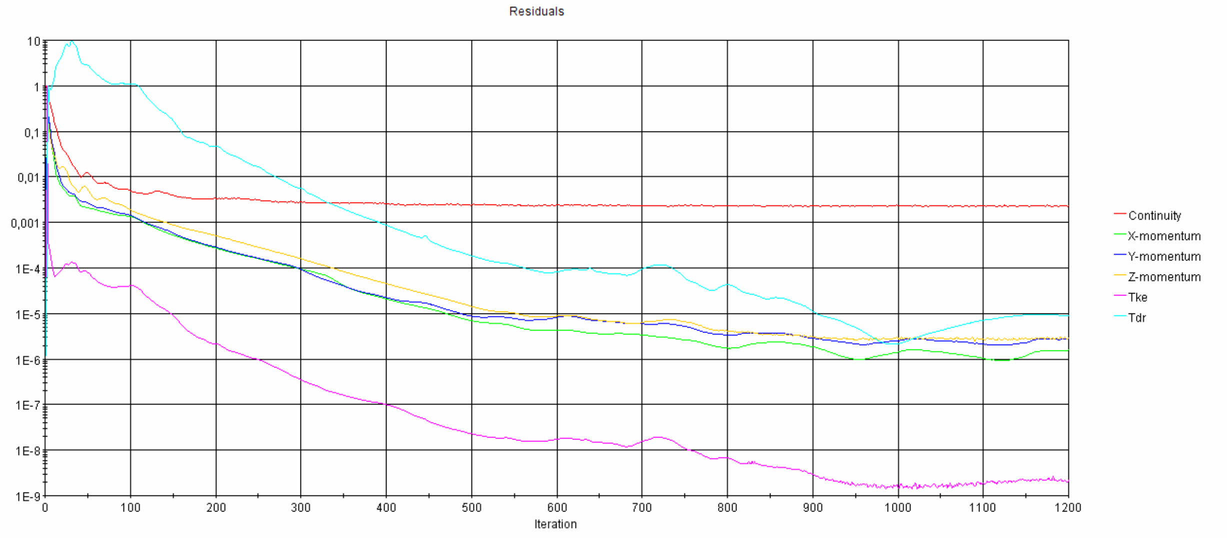
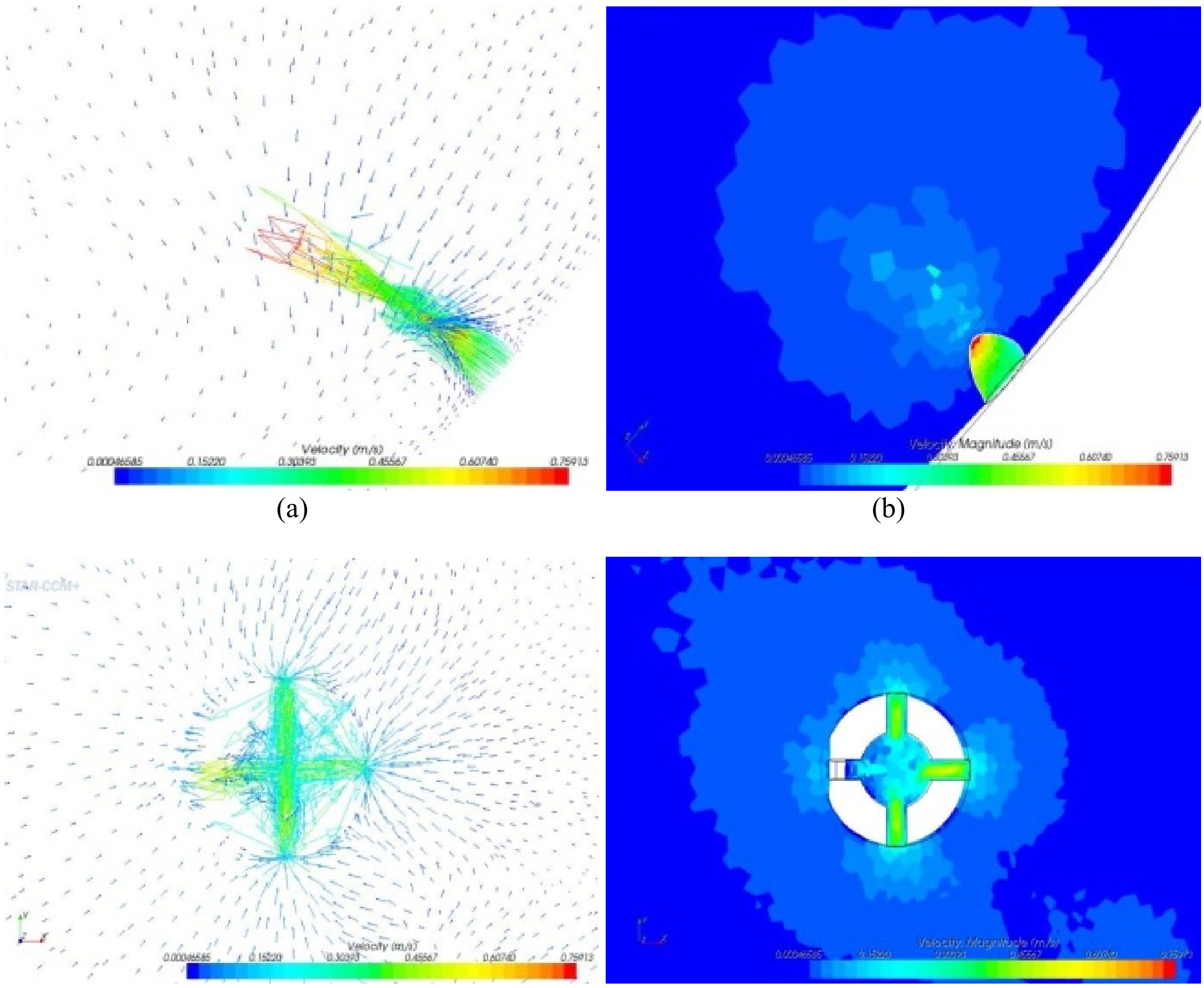


Figure 4. Values of residuals in the model

3.1 Velocity fields on flow pattern

Nowadays, modern Computational Fluid Dynamics (CFD) software helps modelers to define mixing efficiencies for different digester configurations before construction; or to determine possible virtual changes in construction parameters. CFD visualization and analysis also provide an opportunity to examine inlet configurations. Velocity is then considered the more representative parameter of behavior of flow. For this reason the velocity is detailed carried on by defining virtual planes inside the digester (0m and 0.7 m of height) to study the maximum and minimums values, flow direction and turbulent zones. In Figure 5, is visible the details of velocity fields in scalar and vector form in the areas of the nozzles and the central pipe. The blue color represents lower velocities and red color the highest values.

Areas where more mixing is present lay on the height of nozzles, which are responsible for giving the turn to flow, and areas near to the outlet in the center pipe, which dimensions the output is generated turbulence in this area. On the planes at 0m and 0.7m of height, the velocities were mostly of 0.31 m/s and 0.0749 m/s respectively. There has been a maximum speed of 1.4 m/s. Planes of bottom in the digester and 0.7 m. have been considered in all simulations, as depicted in Figures 6 and 7.



(a) (b)

(c) (d)

Figure 5. Detail on velocity fields in significant points of the digester. (a) y (b) nozzles and (c) y (d) centre pipe

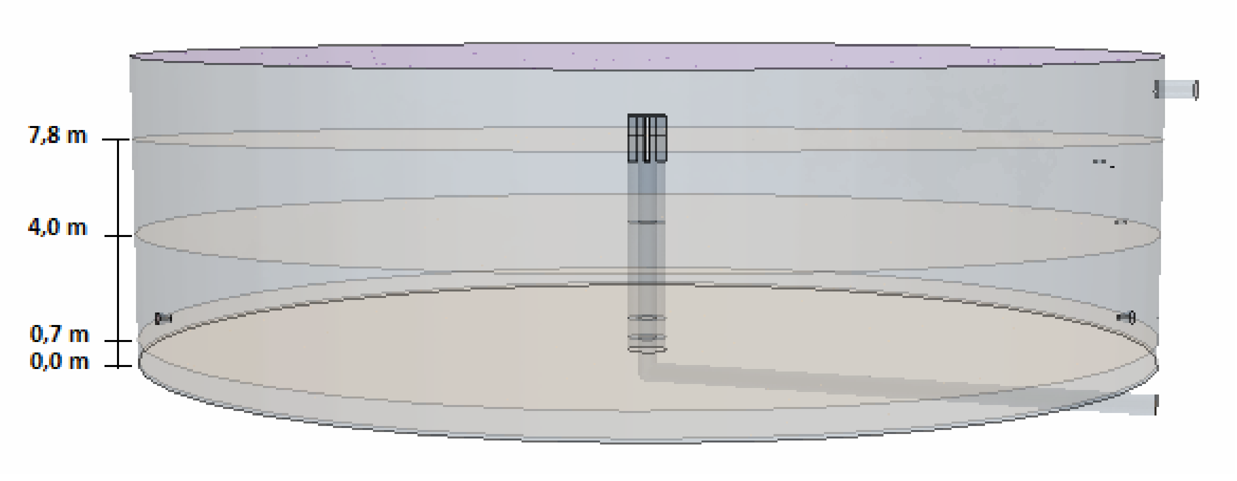
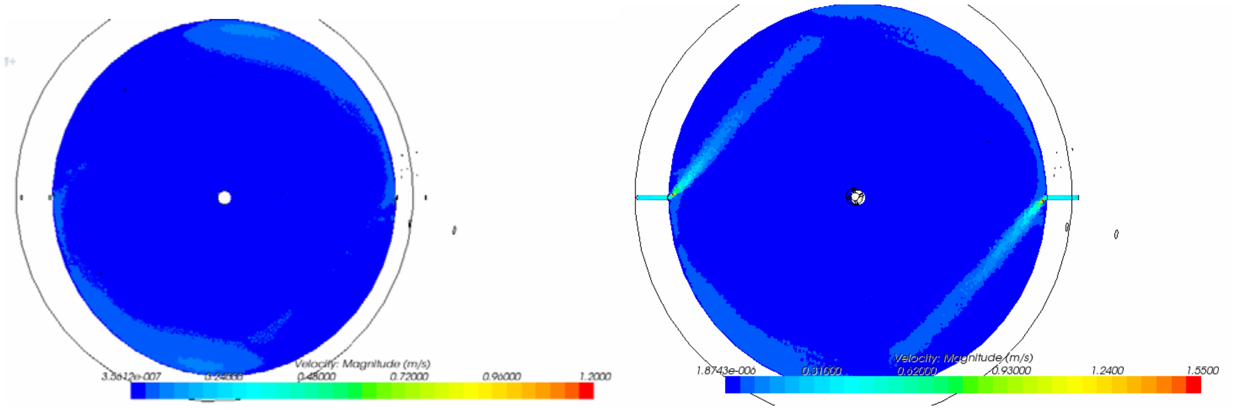


Figure 6. Planes in the simulations considered for analyzing velocities and streamlines



(a) (b)

Figure 7. Detail on velocity fields inside digester on bottom (a) height h=0m and on the nozzle (b) height h=0.7m

3.2 Changing the flow direction

One of the more important parameter determining the inside velocities is the geometry at the entry angle of flow direction on the nozzles; that is defined like α if viewed from the top view of the digester, and β if viewed from the side right face. Four simulations with change of direction flow have been performed: Simulation 1, with angles α=30º and β=11º. Simulation 2, with angles α=0º and β=0º. Simulation 3, with angles α=30º and β=22º. Simulation 2, with angles α=15º and β=0º. On Figure 8, are detailed angles of the nozzles from both perspectives.

All simulations providing a good mixing inside the digester, but some differences were found between the simulations. In cases 1, 3 and 4, the flow follows the course of the shape tank and down a continuous circular movement within the digester. In simulation 2, the flow presented a different behavior, as the shock front is a sweeping flow from the middle of the digester which begins in the nozzle exit, crossing a distance of roughly the distance from the radio and returns.

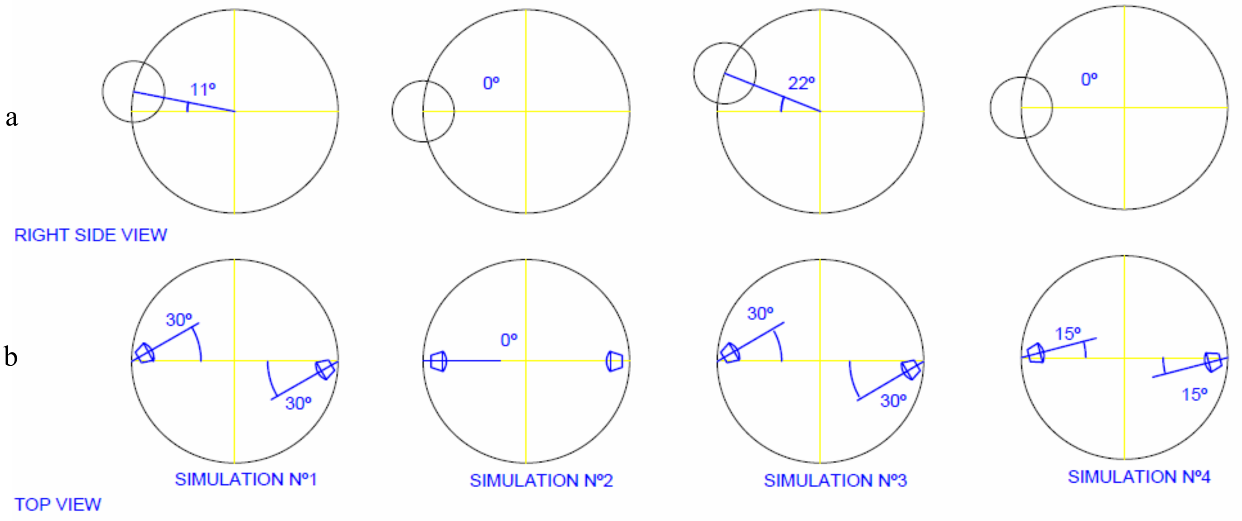
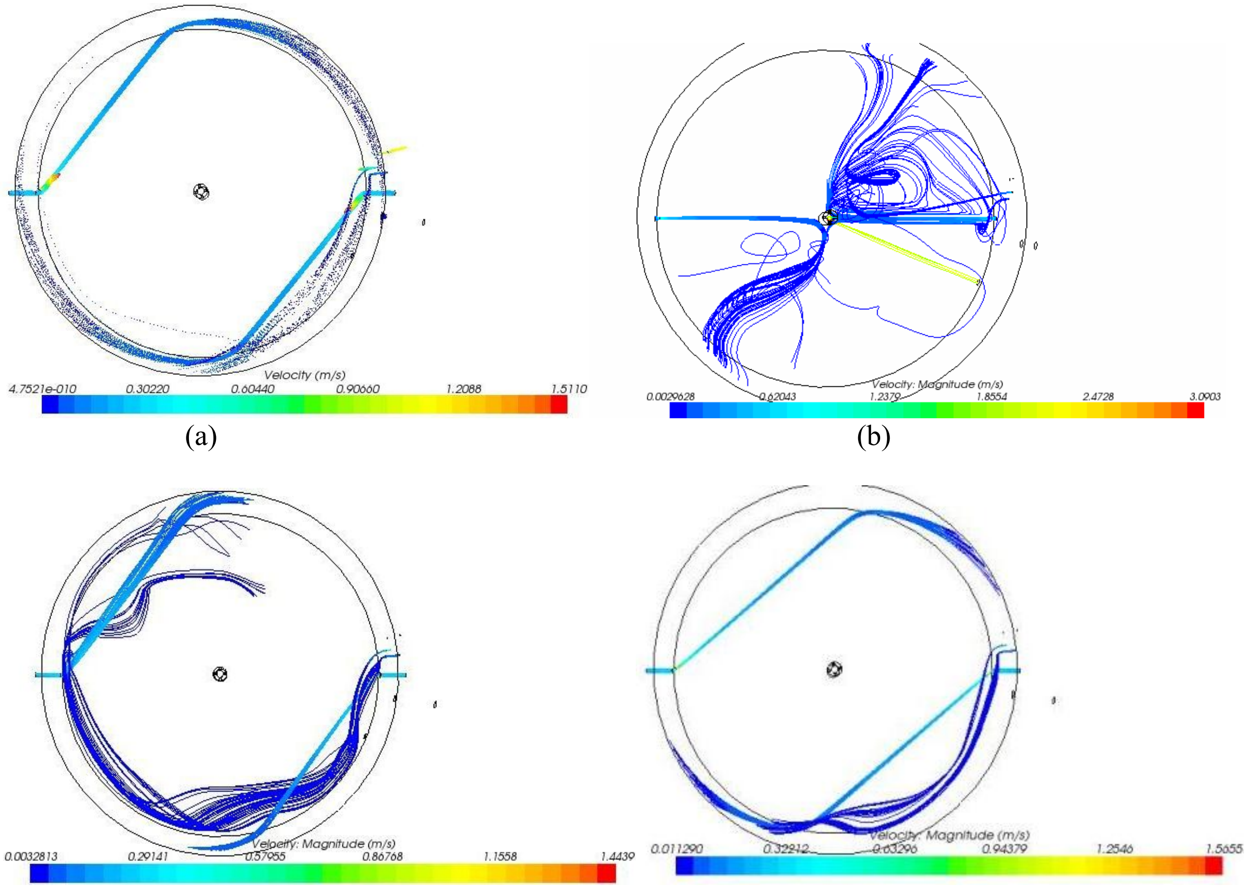


Figure 8. (a) Right side view of simulations. (b) top view of simulations

The stream lines predict the trajectory of starting flow in a determinate moment, which confirms the movement inside the digester. Figure 9 present streamlines in all simulation in the plane of the entrance nozzles (h=0.7m).



(a) (b)

(c) (d)

Figure 9. Stream lines inside digester. (a) Simulation Nº1. (b) Simulation Nº2. (c) Simulation Nº3. (d) Simulation Nº4

3.3 Mixing zones definition

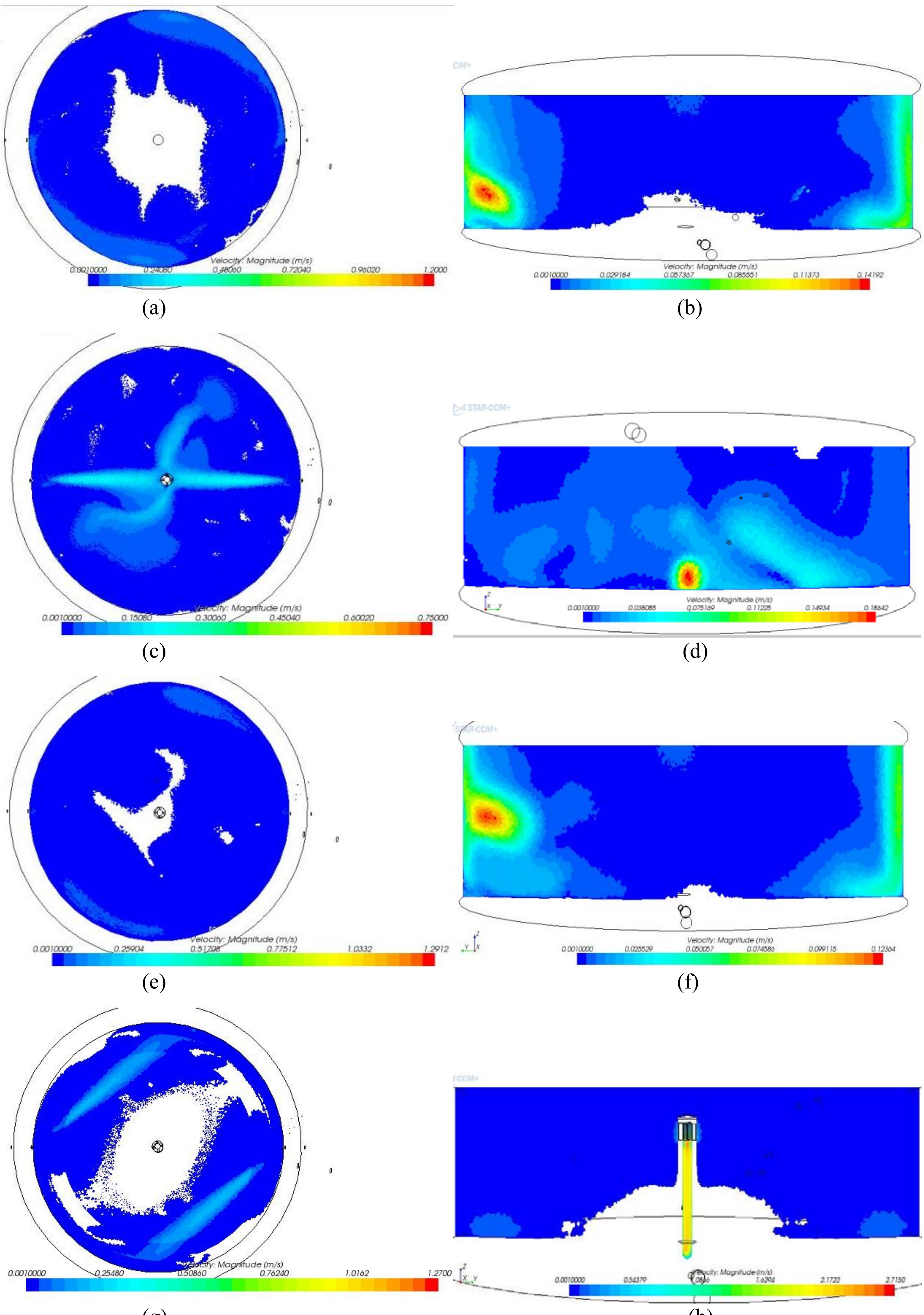
For defining dead zones Stoke´s law for settling has been considered. Velocity flow less the Stoke´s settling velocity of a biomass particle of 245 µm diameter and 1050 kg/m3 bulk density, 0.001 m/s [9, 16] is implemented. Therefore defining velocity limits: Dead zones (velocity <= 0.001m/s) and mixing zones (velocity >= 0.001m/s).

On simulation Nº1, at 0m and 0.7m of height, the 85% of zones are mixing and at 4m and 7.8m the 94%. It shows that the more nixing zone is near to the walls of digester that is stimulated for the angle of nozzles.

The dead zone is in the middle in the bottom of digester. For the simulation Nº2, the values of mixing zones are the 96% at 0m, 98% at 0.7m of height and at 4m and 7.8m the 95%; this simulation has less dead zone on the middle and the wall in the bottom of digester. In the simulation Nº3 the 94% of mixing zones are at 0m and 0.7m of height, and 95% for 4m and 7.8m of height; and the dead zone is located at the same part of the previous simulations.

In simulation Nº4, mixing zones are the 66% at 0m, 83% at 0,7m and to 4m and 7.8m the 100%. In this case the amounts injected flow and does not interact with the sludge that is below nozzles, forming dead zones. The flow when is going up by the walls, begins to mix from the walls, since a total mixture in the upper parts of the digester is finally achieved.

In Figure 10 velocity is represented, mixing zones by visible colour areas and dead zones in blanks spaces, from both perspectives front view and side view. The colour range is the same as is followed in all graphics text.



(a) (b)

(c) (d)

(e) (f)

(g) (h)

Figure 10. Simulation Nº1, at 0m top view (a) and side view (b). Simulation Nº2, at 0m top view (c) and side view (d). Simulation Nº3, at 0m top view (e) and side view (f). Simulation Nº4, at 0m top view (g) and side view (h)

3.4 Evaluation criteria of good mixing

With the results of the simulations, the best mixing flow geometry was identified; by means of percentages of mixing zones analyzed, whereas if all values in the different sections studied (0, 0.7, 4 and 7.8 meters of height) made in the digester are above 85% has a good quality mix or below this percentage has a inappropriate mix. Once the percentages of mixing is analyzed, that matters is the located of dead zone and thus is sensitive to the change in the direction of flow. Simulations number 2 and 3 showed better mixing zones. See Figure 11.

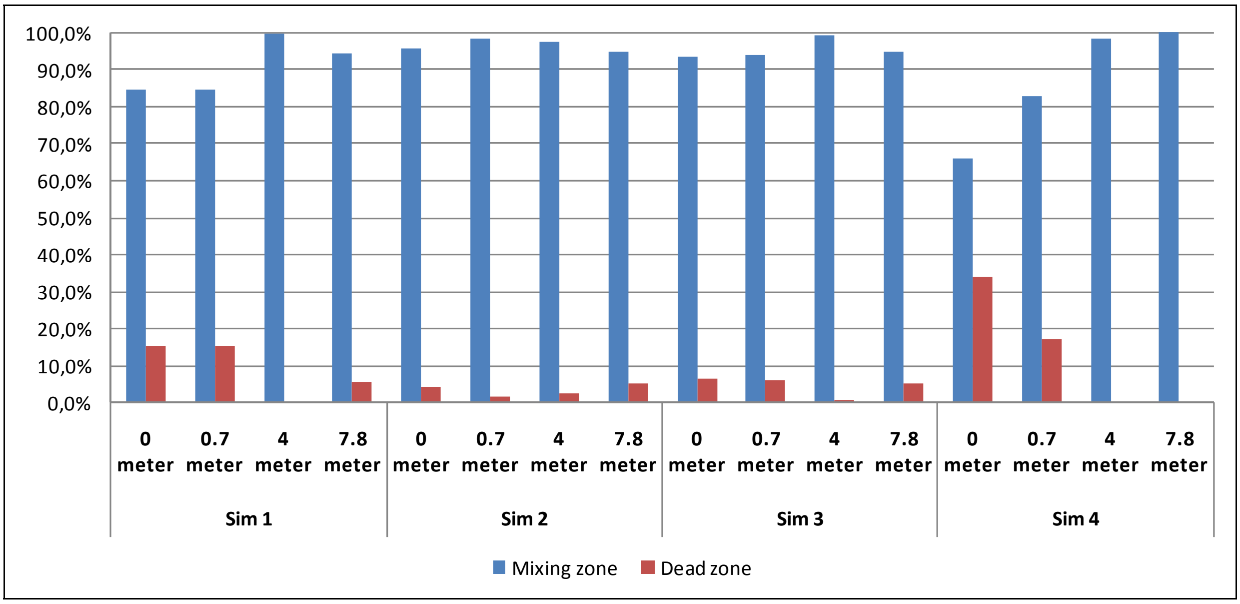


Figure 11. Mixing percentages in each of simulation

4. Conclusion

CFD simulations on an anaerobic digester system have been performed. The applied closures, boundary conditions and models predicted the observed flow pattern in the digester.

Through building a CFD to the flow inside this anaerobic digester, hydraulic behavior of the inside flow has been represented and an interpretation of internal movement of the sludge has been provided.

By means of different simulations, the variability of the data flow changes has been studied, the distribution of velocities and streamlines in the geometry, playing a decisive role as the inflow nozzles, and which may determine the occurrence of dead zones along the flow.

The knowledge of the velocity within the digester as a whole is of paramount importance to identify areas below the threshold velocity of sedimentation of the solids of 0.001 m/s. This supposition has allowed the determination of when the solid particles can be considered as sedimentable.

Thus, most of cells have presented simulated velocities above this value, only some of them are below this velocity.

Different entrance angles of the nozzles directing the flow have been analyzed, determining an optimum direction, when considering mixing in different heights of the digester. Particular geometry of nozzles enhances the digester’s mixing and thus the overall performance. A methodology of quantifying the presence of dead zones has been presented in this paper.

Anaerobic digester is an element of treatment plant completely opaque for managers. Thus, to have a computational model that allows the internal analysis at all points of the geometry is of vital importance for the analysis of the behavior and detection of possible deficiencies. Design optimization based on calibrated models becomes an ideal tool to determine best solutions in engineering proposals for anaerobic digesters.

There is a need for further research and improvement of this model in order to validate the numerical results presented in this paper. Work is ongoing towards this goal.

The results obtained emphasize the importance of considering mixing when simulating anaerobic digestion and, consequently, during reactor design. Decisions must be taken in the design phases of the digester as they affect very much the performance of the whole system. In this sense, CFD techniques postulate as a very promising tool for designers to optimize hydraulic behavior of digesters.

Acknowledgements

Authors want to acknowledge to Entidad Pública de Saneamiento de Aguas Residuales. This work has been partially supported by Instituto de la Mediana y Pequeña Industria Valenciana (IMPIVA, Project IMIDTF/20102/152) and Fondo Europeo de Desarrollo Regional (FEDER), and the Project "Aplicación de técnicas de la mecánica de Fluidos Computacional al Movimiento de Flujos Ambientales" Universitat Politècnica de València research incentives; which are gratefully acknowledged

References

[1] Mehul S. Vesvikar, Muthanna Al-Dahhan. Flow pattern visualization in a mimic anaerobic digester using CFD. Biotechnology and Bioengineering 2005, 89(6), 719-732.
[2] Bello-Mendoza R and Sharratt PN. Modelling the effects of imperfect mixing on the performance of anaerobic reactors for sewage sludge treatment. J Chem Techno Biotechnology 1998, 71:121–130.
[3] Monteith HD, Stephenson JP. Mixing efficiencies in full-scale anaerobic digesters by tracer methods. J WPCF 1981.53:78–84.
[4] Fleming JG. Novel simulation of anaerobic digestion using computational fluid dynamics. PhD Dissertation. North Carolina State University, Raleigh, NC. 2002.
[5] Keshtkar A, Meyssami B, Abolhamd G, Ghaforian H, Asadi MK. Mathematical modeling of non-ideal mixing continuous flow reactors for anaerobic digestion of cattle manure. Bioresour Technol 2003. 87:113–124.
[6] Pena MR, Mara DD, Piguet JM. Improvement of mixing patterns in pilot-scale anaerobic pond treating domestic sewage. Water Sci Technol 2003. 48(2):234–242.
[7] Vesvikar MS, Al-Dahhan MH. Flow pattern visualization in a mimic anaerobic digester using CFD. Biotechnol Bioeng 2005. 89:719–732.
[8] Vesvikar MS, Varma R, Karim K, Al-Dahhan MH. Flow pattern visualization in a mimic anaerobic digester: Experimental and computational studies. Water Sci Technol 2005. 52:537–543.
[9] Karim K, Varma R, Mehul V. M.H. Al-Dahhan. Flow Pattern visualization of simulated digester. Water Research 2004. 38, 3659-3670.
[10] Karim K, Klasson KT, Hoffmann R, Drescher SR, DePaoli DW, Al-Dahhan MH. Anaerobic digestion of animal waste: Effect of mixing. Bioresour Technol 2005. 96:1607–1612.
[11] Karim K, Hoffmann R, Klasson KT, Al-Dahhan MH. Anaerobic digestion of animal waste: Waste strength versus impact of mixing. Bioresour Technol 2005. 96:1771–1781.
[12] Karim K, Hoffmann R, Klasson KT, Al-Dahhan MH. Anaerobic digestion of animal waste: Effect of mode of mixing. Water Res 2005. 39: 3597–3606.
[13] Binxin Wu, Shulin Chen. CFD Simulation of Non-Newtonian Fluid Flow in Anaerobic Digesters. Biotechnol Bioeng 2007. 99:700–711.
[14] USER GUIDE STAR-CCM+ Version 5.02. © 2009 CD-adapco.
[15] Launder B.E., Spalding D.B. (1972). Matematical Models of Turbulence. Academic Press. London,UK Peak RS. X-analysis integration (XAI) technology. Georgia Tech Report EL002-2000A, 2000.
[16] ITT Water & Wastewater. Design recommendation. Digester mixing. http://www.itttreatment.com/designrecommendations/m_digestion_digester_mixing.pdf

Alexandra Martínez Mendoza received her Bachelor's Degree in Environmental Engineering at University of Medellin in 2005. In 2009 she received Master's Degree in Hydraulic and Environmental Engineering from the Universitat Politècnica de València. Currently she is pursuing Doctoral Degree in Water and Environmental Engineering from Universitat Politècnica de València. She is researching about the modeling of flow in anaerobic digester and the modelling of quality water.
E-mail address: almarmen@posgrado.upv.es

Tatiana Montoya Martínez. M.Sc. and PhD in Hydraulic and Environmental Engineering (Universitat Politècnica de València). She received her Bachelor's Degree in Sanitary Engineering at University of Antioquia in 2000. She is head of Research, Development and Innovation Department and Laboratory Coordinator of the Wastewater Area in *Aguas de Valencia* company since 2007. She is author and editor of several publications about Environmental Engineering and Wastewater Management. She has participated in national R&D projects, Seminars and Congress.
E-mail address: tmontoya@aguasdevalencia.es

Vicente Fajardo Montañana. M.Sc. in Environmental Health and Bachelor Degree in Pharmacy. He has more than fifteen years of experience in Wastewater Treatment Plants (WWTP) management and operation. He has managed more than fourty WWTP. He is author and editor of several technical publications about Environmental Engineering and Wastewater management. He has participated in national R&D projects, Seminars and Congress. At the moment, he is the director of Wastewater Area in *Aguas de Valencia* company, which operate more than two hundred plants in Spain.
E-mail address: vfajardo@aguasdevalencia.es

P. Amparo López Jiménez. M.Sc. and PhD in Industrial Engineering, Associate Professor in the Department of Hydraulic Engineering. She is currently the Associate Director of the Hydraulic and Environmental Engineering Department of Universitat Politècnica de València. She has more than a decade of experience in research and teaching in Engineering fields, always related to the hydraulic topics and environmental modelling. She is author and editor of several publications about Hydraulic and Environmental Engineering. She has participated in national and international R&D projects and co-organized International Seminars and Networks. She is an experienced University Teacher, an active researcher and a former practicing engineer.
E-mail address: palopez@upv.es