CHEMICAL ENGINEERING TRANSACTIONS

VOL. 38, 2014

A publication of AIDIC

The Italian Association of Chemical Engineering
www.aidic.it/cet

Guest Editors: Enrico Bardone, Marco Bravi, Taj Keshavarz

ISBN 978-88-95608-29-7; **ISSN** 2283-9216

DOI: 10.3303/CET1438022

Numerical Study of Different Inlet Configurations on the Fluid Dynamics of an Anaerobic Sequencing Batch Reactor

Guilherme Z. Maurina[a], Leonardo M. da Rosa*[a], Lademir L. Beal[a], Ana P. Torres[b] and Maira Sousa[b]

[a]Laboratory of Environmental Technology, University of Caxias do Sul, 1130 Francisco Getúlio Vargas street., Caxias do Sul – Brazil
[b]Research and Development Center, Petrobras, 950 Horácio Macedo avenue, Rio de Janeiro – Brazil
lmrosa1@ucs.br

Concern for the environment is a current trend. Due to problems with the greenhouse effect, biohydrogen emerges as an interesting source of energy: its combustion does not emit carbon dioxide into the atmosphere, and its production by fermentative routes also contributes to the environment by the consume of wastes.
In this work, Computational Fluid Dynamics (CFD) technique is applied to study the flow in a bioreactor used for biohydrogen production, using different distributors for the injection of the recirculating liquid mixture inside the reactor. Inlet configurations with seven and nineteen pipes, direct downward and upward, were compared to the original configuration, with one pipe. Results indicate that the use of a distributor with a larger number of pipes provides higher turbulence inside the reactor, when directed downward. This means a better mixture of the phases, which can improve the reactor efficiency.

1. Introduction

Fossil fuels have played a fundamental role in the industrial development and are responsible for fulfilling 80 % of energy demand globally. However, this energy system is now facing two fundamental problems: gradual depletion and environmental pollution. This lack of sustainability has led to extensive research on new alternative energy sources (Brey et al., 2006).
Among various alternative energy sources, biohydrogen is regarded as the most promising future energy carrier, due to its low cost and ability to transform waste into environmentally friendly materials (Jung et al., 2011). The main challenges are the low hydrogen yield and production rate which can be improved by determining the optimal bioreactor design (Show et al., 2011).
For a successful design and scale-up of a bioreactor, understanding the inlet configuration influence on the flow is crucial. The inlet stream can be used to promote mixture in the reactor. The promotion of mixing is critically important to ensure the availability of nutrients and other essential substances to the growing cells (Bannari et al., 2011). Poor distribution of the flow is generally caused by imprecise design and fabrication of the distributor, which will generally increases back-mixing and decreases the driving force of mass/heat transfer (Fan et al., 2009).
Recent advances in CFD modelling and availability of low cost and high speed computers have allowed performing three-dimensional simulations of complex multiphase flows. CFD strategy already developed for simpler gas-liquid vessels can be applied to the design of bioreactors (Montante et al., 2013).
In this work, the OpenFOAM 2.2.0 is used, to simulate the gas-liquid flow in a bioreactor used for biohydrogen production. The aim of this paper is to evaluate the effects of different inlet configurations on mixture inside an anaerobic sequencing bath reactor (ASBR), which can affect hydrogen production.

Please cite this article as: Maurina G., Rosa L., Beal L., Torres A., Sousa M., 2014, Numerical study of different inlet configurations on the fluid dynamics of an anaerobic sequencing batch reactor, Chemical Engineering Transactions, 38, 127-132 DOI: 10.3303/CET1438022

2. Methodology

An pilot scale ASBR with internal diameter of 0.604 m, height of 3.8 m and a total capacity of 1,000 L has been employed. To study the influence of the inlet configuration on the mixture inside the reactor, a total of five configurations were evaluated. The first (Case A) considered one pipe, located at the center of the bottom of the bioreactor. This configuration, which has a conical region at the base to minimize the presence of “dead zones”, is already operating in a pilot plant for biohydrogen production. Cases B and C considered seven pipes directed upward and downward respectively. In Cases D and E nineteen pipes were considered, directed upward and downward respectively. In all cases, the inlet and outlet pipes had 40 mm of diameter. Figure 1 illustrates these configurations, and Table 1 summarizes the simulated cases. The gas outlet is placed at the top of the reactor, and the liquid recirculation pipe is located at its lateral surface.

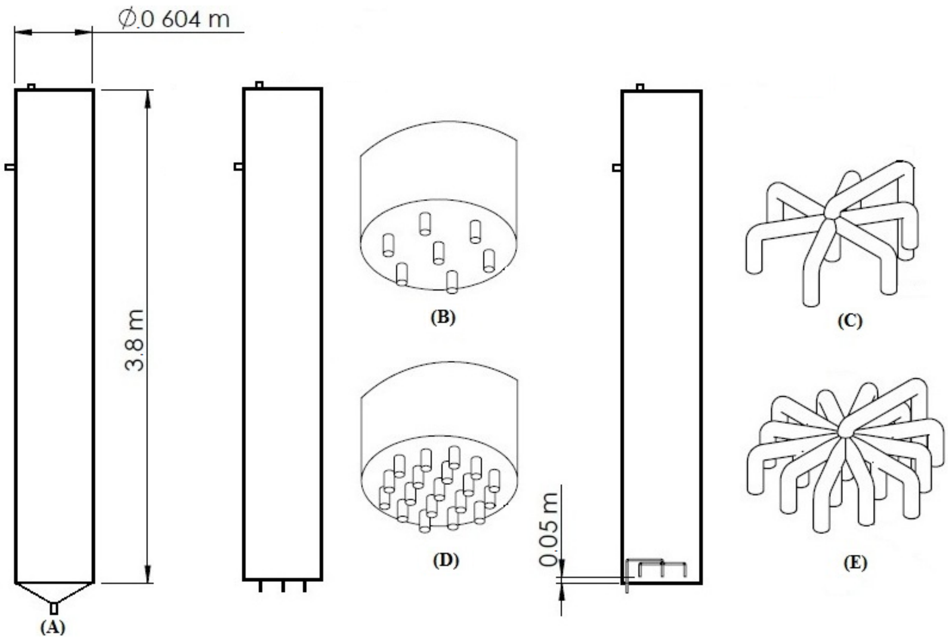

Figure 1: Schematic view of the bioreactor and the inlet configurations used in Cases A, B, C, D and E.

Table 1: Characteristics of inlet configurations used in Cases A, B, C, D and E.

Case	Number of inlet pipes	Direction
A	1	up
B	7	up
C	7	down
D	19	up
E	19	down

Tests were performed using seven mesh sizes ranging from 60,000 to 210,000 control volumes in a previous study (Maurina et al., 2013), and it was verified that a mesh with approximately 130 thousand control volumes is sufficient to predict accurately the flow in this bioreactor. Higher refinement was applied both near the inlet regions, to capture correctly the effect of the different distributors on the flow, and near the outlet regions, which can present higher velocity gradients. Due to differences in the geometries, small changes were necessary to adapt the mesh to each case.

A three-dimensional, transient, Eulerian-Eulerian two-phase fluid model was adopted to describe the flow behavior of each phase. Thus, gas phase (biogas) and liquid phase (a mixture of substrate and activated sludge) are treated as different continua.

The coefficient of the drag force exerted by the gas phase on the liquid phase was obtained using the Zhang and Vanderheyden correlation. The lift force coefficient was calculated using the Legendre and Magnaudet correlation. The virtual mass force was also considered, using a constant value of 0.5 for its coefficient (Auton et al., 1988). The k-ϵ turbulence model (Launder and Spalding, 1973) was applied to account for the turbulent behavior of the mixture.

The recycling flow rate was maintained constant, equal to 2.25 L/s. In order to maintain the volume of liquid constant in the reactor, a function was applied to calculate the amount of liquid that leaves the reactor through the lateral pipe, and use it as an inlet condition,

$$\alpha_{g,in} = 1 - \frac{U_{l,out}\, A_{out}\, \left(1 - \alpha_{g,out}\right)}{U_{l,in}\, A_{in}} \tag{1}$$

where U_l is the liquid velocity (m s^{-1}), α_g is the gas volume fraction (m^3 m^{-3}) and A represents the boundary area (m^2). Other conditions applied to carry out the simulations are given in Table 2. In all cases, gas is injected through the entire base of the reactor, to make the distribution of the bubbles as much uniform as possible, as expected in a bioreactor. A gas flow rate of 0.0925 L/s, which corresponds to the biogas production in the pilot bioreactor, was applied.

Table 2: Operational conditions and physical properties used in the numerical simulations.

	Operational Conditions	
Inlets	Liquid flow rate of 2.25 L/s Gas flow rate of 0.0925 L/s	
Outlet	Pressure of 1 atm	
Walls	Smooth surface, non-slipping condition for both phases	
	Physical Properties	
Phase	Density	Kinematic Viscosity
Disperse (gas)	0.089 kg/m³	8.4×10^{-6} m²/s
Continuous (liquid)	1,009.7 kg/m³	1.0×10^{-6} m²/s
Bubbles Diameter	1 mm	

The coupling between the pressure and velocity fields is done according to a mix between the SIMPLE and PISO algorithms. The total simulation time for each simulation was 300 s. There was verified that 200 s are needed to stabilize the flow, comparing average results obtained with different flow times. Thus, only the last 100 s were used to calculate average values. During the simulations, as convergence criterion the total residue was kept below 10^{-5}, and as stability criterion the Courant-Friedrichs-Lewy (CFL) condition of less than one was applied:

$$CFL = U \frac{\Delta t}{\Delta x} \tag{2}$$

where U is the mixture velocity (m s^{-1}), t is the time step (s) and x is the control volume size (m).

3. Results and Discussion

Simulations were carried out, and average values were collected. Average values are presented in the figures below, which show transversal sections of the reactor along its height. It should be noticed that not all of the inlet pipes of each case can be seen in these transversal sections. Liquid velocity fields obtained in all cases can be seen in Figure 2. In the simulation of Case A (Figure 2A), the use of only one inlet pipe resulted in an inlet stream with higher velocity (2.07 m/s) at the bottom of the reactor. It should be noticed that even though the inlet configuration is symmetric, turbulence inside the reactor causes asymmetries in the flow. The use of distributors (Figures 2B, 2C, 2D and 2E) resulted in a more uniform velocity distribution. The use of more inlet pipes also caused the liquid to be fed in the reactor with a lower velocity, in order to maintain the same inlet flow rate. Nonetheless, it is interesting to note that Case E, with nineteen pipes, resulted in values of liquid velocity as high as 0.2 m/s along the center of the reactor (Figure 2E).

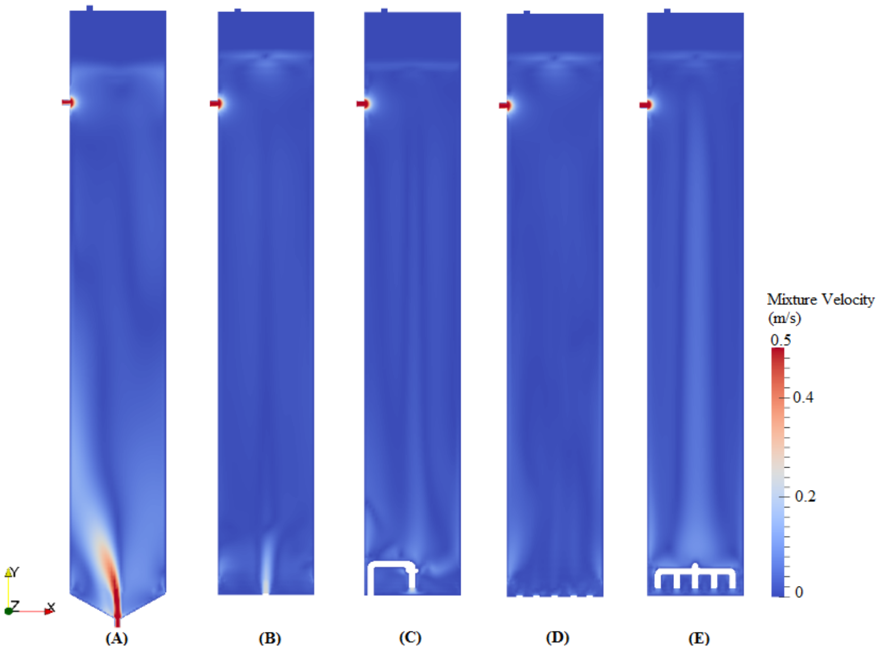

Figure 2: Average axial liquid velocity obtained in Cases A, B, C, D and E.

Figure 3 shows details of the bottom region of each case, with vector fields of the liquid velocity. Figure 3A shows the higher velocity of the inlet stream present in Case A, which dominates the flow in this region. Case B, which considered a distributor with seven pipes direct upward, is shown in Figure 3B. Liquid velocity is higher at the pipe shown, and the other inlets caused the presence of vortices in the average liquid velocity field. In Case C, there are seven pipes directed to the bottom of the reactor (Figure 3C). This caused the flow to be more uniform overall, except near the distributor walls. Nineteen pipes directed upward were used in Case D, which resulted in the most uniform velocity field, near the inlet zone, as can be seen in Figures 2D and 3D. Figure 3E presents the vector field obtained using nineteen pipes directed downward (Case E). Different from Case D, in this case the velocity field is not uniform: large vortices can be seen above the distributor structure. In Case E, it is shown that the distributor structure affects strongly the flow pattern.

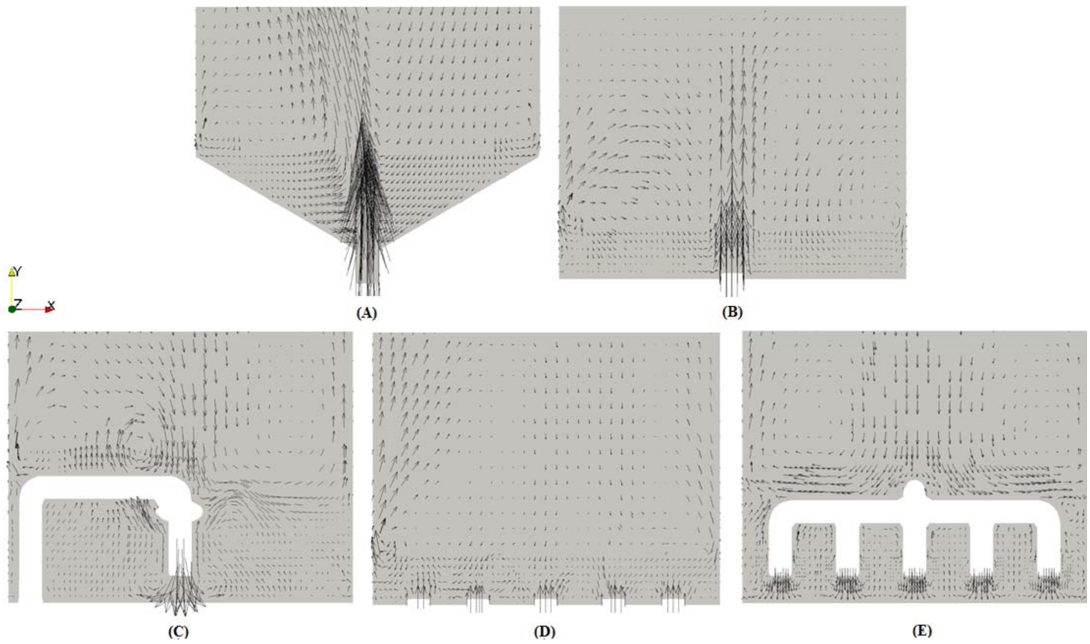

Figure 3: Vector fields at the inlet region obtained in Cases A, B, C, D and E.

The average turbulent kinetic energy (k) fields obtained for each case are presented in Figure 4. As expected, the higher velocity found in Case A resulted in higher turbulence near the inlet region, which is propagated through the reactor. Higher values of k were also observed near the top of the reactor, close to the liquid free surface, and at the lateral outlet, for all cases. In this region, liquid accelerates due to the recirculation pipe. The use of distributors directed upward (Figures B and D), which caused a more uniform liquid velocity, resulted in lower values of turbulence. In Figure C, it can be seen that the use of seven pipes directed to the bottom of the reactor resulted in an almost uniform field for turbulent kinetic energy, except for the higher values found near the top of the reactor. This is the same behavior found using nineteen pipes (Figure 4E), which resulted in higher values for turbulence throughout the reactor.

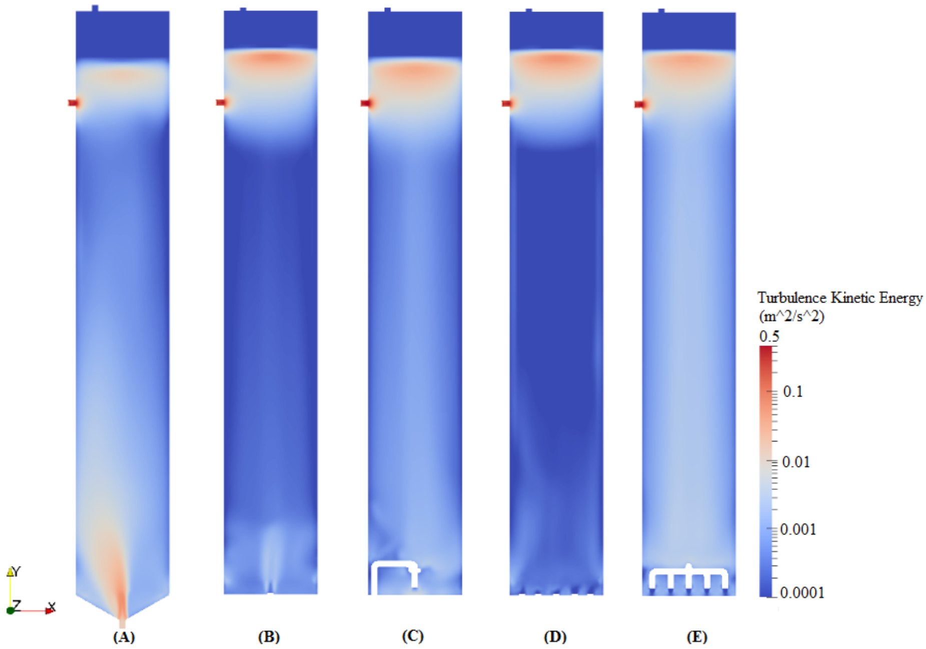

Figure 4: Average turbulent kinetic energy fields obtained in Cases A, B, C, D and E.

Mean values for turbulent kinetic energy were calculated for each case, weighted by the liquid volume present in each control volume, according to the equation:

$$\bar{k} = \frac{\sum k\,\alpha_l V}{\sum \alpha_l V} \tag{3}$$

where k is the turbulence kinetic energy, α_l is the liquid volume fraction and V is the volume of each control volume.

Figure 5 presents the values obtained using Equation 3 for each case. Case A resulted in a mean value of 0.0014 m^2/s^2. The use of distributors direct upward (Cases B and D) resulted in approximately 0.0011 m^2/s^2 of mean turbulent kinetic energy. This represents nearly 20 % of reduction of the turbulence inside the reactor. Case C resulted in a flow as turbulent as in Case A. In Case E, the calculated mean turbulent kinetic energy was 0.0018 m^2/s^2. This indicates that, using this distributor, the turbulence can be improved with 27 %.

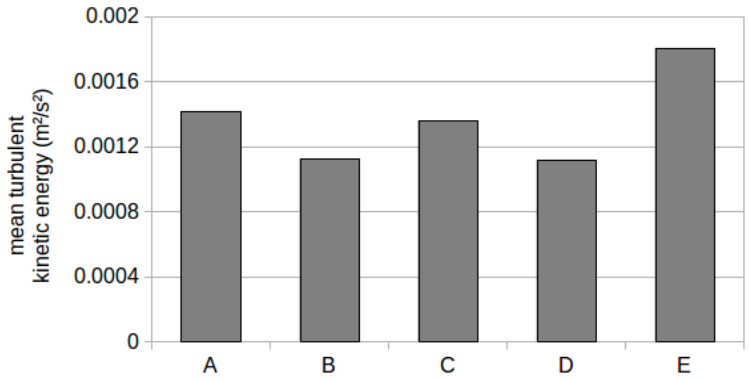

Figure 5: Mean turbulent kinetic energy values calculated for Cases A, B, C, D and E.

4. Conclusions

Simulations of gas-liquid flow in a bioreactor with different inlet configurations were carried out in this work. Results showed that the use of distributors can affect the flow field. The turbulence throughout the reactors evaluated showed a dependency on the inlet configurations. The use of more inlet pipes, directed upward, provided a more uniform flow, but also resulted in a less turbulent flow field, which can cause a poor mixture inside the reactor.

When considering distributors directed to the bottom of the reactor, the use of a distributor with seven pipes resulted in a flow as turbulent as the original case, with only one inlet. A configuration with nineteen pipes resulted in the most turbulent flow field, indicating that the use of a distributor with large number of pipes direct downward can improve the mixture of the phases in this bioreactor.

Acknowledgements

This research was financially supported by PETROBRAS and FAPERGS.

References

Auton T.R., Hunt J.C.R., Prud'homme M., 1988, The force exerted on a body in inviscid unsteady nonuniform rotational flow, J. Fluid Mech 197, 241-257.

Bannari R., Bannari A., Selma B., Proulx P., 2011, Mass transfer and shear in an airlift bioreactor: Using a mathematical model to improve reactor design and performance, Chemical Engineering Science 66, 2057-2067.

Brey J.J., Brey R., Carazo A.F., Contreras I., Hernandez-Diaz A.G., Castro A., 2006, Designing a gradual transition to hydrogen economy in Spain, Int. J. Hydrogen Energy 159, 1231-1240.

Fan Z., Zhou X., Luo L., Yuan W., 2009, Numerical investigation of constructal distributors with different configurations, Chinese Journal of Chemical Engineering 17, 175-178.

Jung K.W., Kim D.H., Kim S.H., Shin H.S., Bioreactor design for continuous dark fermentative hydrogen production, Bioresource Technology 102, 8612-8620.

Launder B.E., Spalding D.B., 1973, The numerical computation of turbulent flows, Computers Methods in Applied Mechanics and Engineering 3, 269-289.

Maurina G.Z., Rosa L.M., Beal L.L., Baldasso C., Pederiva L., Torres A.P., 2013, Optimization of a hydrogen production bioreactor using computational fluid dynamic (CFD) techniques, Proceedings of the 13th World Congress on Anaerobic Digestion, Santiago de Compostela, Spain, 25-28 June.

Montante G., Coroneo M., Francesconi J.A., Paglianti A., Magelli F., 2011, Computational analysis of a vortex ingesting bioreactor for hydrogen production 32, 721-726.

Show K.Y., Lee D.J., Chang J.S., 2011, Bioreactor and process design for biohydrogen production, Bioresource Technology 102, 8524-8533.