Investigating Wastewater Treatment in MBRs Using Computational Fluid Dynamics

Mitra Bayat[1], Mohammad Reza Mehrnia[2*], Navid Mostoufi[3], Mehdi Rajabi Hamaneh[4]

1. Master Student of Chemical Engineering, Chemical Engineering Department, College of Engineering, University of Tehran, Iran (mitra.bayat@ut.ac.ir)
2. Associate Professor, Chemical Engineering Department, College of Engineering, University of Tehran, Iran
3. Professor, , Chemical Engineering Department, College of Engineering, University of Tehran, Iran (mostoufi@ut.ac.ir)
4. Assistant Professor, Chemical Engineering Department, College of Engineering, University of Tehran, Iran (rajabi@ut.ac.ir)

Extended Abstract

Introduction

Membrane bioreactor (MBR) is an effective technology for wastewater treatment and water reuse which is becoming increasingly popular due to its numerous applications and advantages in conventional activated sludge process. This novel technology have advantages of small footprint, high concentration of mixed liquor suspended solids (MLSS), high removal efficiency of chemical oxygen demand (COD), less production of excess sludge and to be reliable and simple for operation. Membrane fouling and its consequences, regarding plant maintenance and operating costs, has gained attention in the recent years as a major obstacle for development of this technology. Various methods have been used to reduce membrane fouling and new solutions are also frequently proposed and used.

Among different operational variables, aeration is the most effective factor on membrane fouling mitigation. Despite its major role in membrane fouling reduction, the energy consumption of aeration is the main operating cost for MBRs, such that approximately 30-50% of consumed energy in a submerged MBR is used for aeration. Hence, operation improvement by optimizing hydrodynamic conditions has a high technical and economic significance.

Computational fluids dynamics (CFD) is a powerful tool for understanding the relationship between hydrodynamics and fouling in MBRs. Researches have been conducted to assess hydrodynamic and its effect on the system efficiency. Most of design, operational and geometrical variables, like bubble diameter, membranes distance, presence of baffles and walls in flat-sheet modules require evaluation and optimization. Membranes are mostly assumed to be rigid in CFD simulations of MBRs.

In this study, effects of hydrodynamic characteristics of a submerged membrane bioreactor on membrane fouling were investigated using computational fluid dynamics simulation. The effects of hydrodynamic characteristics on fouling in an airlift MBR was also investigated using CFD simulation. Three-dimensional two and three-phase simulation was implemented using Eulerian approach and k-ε turbulent model. Results indicated that by increasing air flow rate and MLSS concentration, shear stress on membrane surface increase and membrane fouling decreases. Furthermore, effect of considering population balance model in simulation was also studied. In addition, the results also indicated that using granular model in three-phase simulation would lead to a more realistic simulation results. Simulation results were in good agreement with experimental data which demonstrate the ability of this CFD approach and population balance model as an efficient tool.

Materials and Methods

Experiments were carried out in a submerged membrane bioreactor which is 70 cm in height, 23 cm in length and 21 cm in width with operating volume of 20 L for activated sludge. A flat-sheet chlorinated polyethylene membrane with mean pore size of 0.45 μm and effective membrane area of 0.11 m^2 was used. Two baffles were located at both sides of membrane. The required air was pumped through a sparger located beneath the membrane, and its flow rate was measured using a flow meter. The biomass was obtained from a municipal wastewater treatment plant in Tehran, Iran. The driving force for filtration was created by vacuum. In all experiments, the system was fed by a synthetic influent, glucose, ammonium sulfate, and ammonium phosphate

---

which are the sources of carbon, nitrogen, and phosphorus, respectively. The tests were carried out at four different air flow rates of 0.2, 0.4, 0.6 and 0.8 m³/h and at two MLSS concentrations of 8 and 12 g/L. The permeate flow rate, MLSS concentration and membrane resistance were measured. Total gas hold-up was also estimated by visual determination of bed expansion.

A three-dimensional two- and three-phase model was used to investigate the hydrodynamics of the MBR. In order to describe liquid and gas properties in multiphase flow, an Eulerian-Eulerian approach was implemented. The standard k-ε turbulent model was used for phases to model turbulence. A fine mesh was generated between the membranes. However, to decrease the number of computational cells, only a quarter of the set-up was considered as the simulation domain due to the symmetry from both sides. The projected area of the air spargers at the bottom of the system were considered as the air velocity inlet boundary condition. The boundary condition at the top of the MBR was set to open to atmosphere in order to let the air exit to the atmosphere. A bubble size distribution with ten bubble classes and the possibility of coalescence and breakage was also used in some of the simulations. In this work, phase-coupled simple with pressure based solver was applied for the Eulerian multiphase simulations. The velocities were solved coupled by phases, but in a discrete method. This method solves momentum and pressure based on continuity equations simultaneously, thus the rate of convergence is improved compared to the segregated method which solves the governing equations sequentially.

Results and Discussion

- The effect of bubble size distribution in MBR

Due to importance of bubble characteristics and their distribution in bioreactors, a simulation was performed to employ the bubble size distribution, based on experimental data. MBR was simulated at TMP of 40 kPa, MLSS concentration of 8 g/L and in four different air flow rates of 0.2, 0.4, 0.6 and 0.8 m³/h. Thus, up to 10 bubble classes, with the possibility of accumulation and breakage, are studied.

Bubble size distribution in 5 stages of bioreactor was investigated. Results indicated that by increasing the aeration intensity, ratio of larger bubbles increases in the system. After formation of air bubbles at the sparger, their coalescence and breakage occur during their movement towards the free liquid surface and gradual increase of bubble diameter can be observed. Larger bubbles are commonly seen at higher levels, near the membrane surface and the wall. Average bubble diameter from both simulation and experimental results were compared. Simulation results are correlated with experimental data which verified bubble size distribution in simulation. Therefore, a mean bubble diameter was used in other following simulations.

- The effect of MLSS concentration and aeration intensity

In order to investigate the effect of aeration as the main effective factor in membrane fouling reduction, simulation was done at various air flow rates of 0.2, 0.4, 0.6 and 0.8 m³/h. Gas holdup was measured experimentally and was compared with simulation results. Results demonstrate that increase in the aeration intensity and consequent growth in average bubble diameter causes a greater gas holdup in the bioreactor. Although the growth in the average bubble diameter leads to reduction of the gas holdup due to higher rise of velocity, the overall effect of increase in the aeration intensity and average bubble diameter is higher gas holdup in the system. Moreover, with as increase in MLSS concentration and consequence increase in the activated sludge viscosity, more bubbles are trapped in the riser which leads to more gas holdup.

In airlift bioreactors, flow of air is the main cause of liquid motion and circulation. Therefore, with increase in the aeration intensity, the liquid velocity is increased in both riser and down comer which leads to a greater shear stress on the membrane surface. Furthermore, at higher MLSS concentrations, which correspond to greater liquid viscosity, air and liquid shear rates are increased. However, at lower aeration intensities, change in the MLSS concentration does not make a significant change in the shear stress. This is due to the fact that aeration cannot impose the necessary rate of mixing in the bioreactor and provide the force required for particles movement. Exerting more shear stress on membrane surface in higher aeration intensity leads to a decrease in cake formation on membrane surface and membrane fouling resistance.

The gas shear stress contours on membrane surface at various air flow rates was also investigated. It was seen that a greater shear stress is exerted on the surface in the middle and upper half of the membrane which is owing to higher velocity and turbulence of gas and liquid mixture in this region. At air flow rate of 0.2 m³/h, the maximum shear stress is exerted on a small part of the membrane surface, while by increasing the air flow rate to 0.8 m³/h a greater surface area is exposed to the maximum shear stress.

- Validation of the model

In order to validate simulation results, gas shear stress and its effect on MBR operation and membrane resistance was studied under different conditions. Results indicate that by increasing air flow rate, resistance reaches its

lowest amount. By increasing aeration intensity stress changes resulting from gas and liquid becomes ascending. It should be mentioned that both stresses influence cake formation and total resistance on surface tension, but it cannot be specifically said which effect is more. Other studies indicate that in constant pressure systems much change is not observed after reaching semi-constant condition. As mentioned before, aeration causes cross flow on membrane in air lift bioreactor and the more aeration causes more flow circulation velocity and lifting force on particles. This leads to membrane resistance reduction. Liquid shear stress changes on membrane surface in different air flow rates and, thus, shows a similar trend.

The gas hold up was also measured experimentally and was compared with the simulation results. It was observed that the simulation results are in good agreement with experimental data which indicates model accuracy and ability of computational fluid dynamics for investigation and prediction of bioreactor hydrodynamics.

The effect and behavior of solid particles distribution

Three-phase simulation was studied in order to approach real conditions and identification of solid particles aggregation. Three-phase simulation was conducted in aeration intensity of 0.8 m^3/h and MLSS concentration of 8 g/L. Average particle diameter was determined by microscopic image of active sludge and image analysis of 6 μm. Eulerian approach was used to model the three-phase simulation.

Volume fraction distribution of solid phase particles of sludge, liquid and air were investigated. Results show that solid particles are accumulated less near membrane and are accumulated more in bottom of bioreactor due to more aeration and liquid circulation around baffles. In order to achieve more uniform distribution of solid particles, air distributor can be placed at the bottom in order to prevent particle accumulation in that area.

Conclusion

A submerged membrane bioreactor was investigated using CFD simulation. A two- and three-phase simulation was implemented using Eulerian approach. In addition, the effect of permeate flux was considered in simulation. Simulation results were validated against the experimental data. From the results reported here, the following conclusions can be drawn:

- By using bubble size distribution, bubbles behavior during their movement in system can be investigated. Results show that bubble size usually increases during their movement from sparger to free surface of liquid and bigger bubbles tend to accumulate near membrane surface and walls.
- By increasing aeration intensity and MLSS concentration in system, the gas and liquid shear stress increases on membrane surface.
- Simulation results were in good agreement with experimental data which indicates model accuracy and ability of computational fluid dynamics for investigation and prediction of bioreactor hydrodynamic.
- Application of granular model in three-phase simulations causes reactor conditions come closer to actual one. Results indicate that solid particles tend to be accumulated more in bottom of bioreactor and less near membrane and around baffles.

Keywords: computational fluid dynamics, fouling, membrane bioreactor, Wastewater treatment, water reuse.