

INFLUENCE OF GEOMETRICAL AND OPERATIONAL PARAMETERS ON THE PERFORMANCE OF BUBBLE REACTOR WITH IMMERSED MEMBRANE MODULE

Dessislava Moutafieva*, Veselin Iliev

University of Chemical Technology and Metallurgy, 8 blvd. Kl. Ohridski, Sofia, 1756, Bulgaria

Abstract

The objective of this work was to study and numerically evaluate the effect of various operation conditions like location of immersed membrane module and aeration design on the hydrodynamics in a pilot scale bubble column with a membrane module. The effects of the membrane module orientation can be more clearly demonstrated by comparing the gas holdup at different cross-sections along the column height and analyzing the shear stress distribution on the membrane surface by CFD. However, the simulation of the flow behavior surrounding the membranes requires high mesh resolutions and hence a large number of elements. The numerical simulation was done with Ansys CFX, using the $k-\epsilon$ model with scalable wall function for turbulence, high resolution advection scheme and second order backward Euler transient scheme for solving the Navier-Stokes equations. The influence of the flow on sludge accumulation is estimated by shear strain rate of the phases. A CFD model was developed which accounts for aeration and membrane configurations. The numerical results of the velocity, pressure and volumetric fraction distributions for the simulations at different geometric characteristics of the membrane are presented and analyzed.

The research demonstrates the capability of using CFD to optimize the design of the membrane module, aeration system and membrane bubble column.

Keywords: Membrane reactors, CFD simulation, flow structure, membrane module design

1. INTRODUCTION

In recent years membrane bioreactors (MBRs) are being more and more used as an alternative to the conventional methods for treating waste water. Currently, membrane bioreactors are being widely used for biological waste water treatment [1], drinking water production [2] and processing of beverages [3].

The hydrodynamics of the membrane reactors and the bubble columns is quite similar. The difference between the two constructions is the presence of a membrane in the membrane reactors which helps in the separation process of the clean water. The placement of a membrane inside the column affects the behaviour of the bubbles [3-6]. The hydrodynamic conditions within submerged MBRs highly depend on the design and the geometry of the module which are of a great importance in reducing sludge deposition on the membrane surface, thus reducing the possibility of membrane fouling. The fouling of the membrane, necessitating frequent cleaning and replacement of the membrane module, is the main drawback of MBR systems. This constricts the application of MBRs in full-scale [6,7] (Ratkovich), and their wide-spread commercialisation due to reduced productivity and increased operational and maintenance costs. The main reason for membrane fouling is the attachment of suspended solids and soluble substances on the membrane surface.

However, submerged MBRs bring several advantages, like lower energy consumption, and smaller physical footprint. In addition, submerged MBRs maintain a more sterile environment, preventing contamination. The problem of membrane fouling is reduced in submerged MBRs due to the presence of gas and the collision of the gas bubbles with the surface of the membrane. A comprehensive review on the effects of aeration on submerged MBRs has reported that membrane performance was enhanced by gas sparging and has shown improvements up to 63% [7-9].

The optimal operation of the membrane bioreactor is related to:

- the position and orientation of the membrane module in the reactor;

- gas / liquid ratio;
- transmembrane pressure;
- sufficient shear stresses in order to favour friction on the membrane surface.

The connection of the latter with the reduction of the resistance around the membrane and the increase of the permeate flow is confirmed qualitatively.

The design of MBRs is significantly influenced by the rate of mixing, the mass transfer coefficient the [10,11] cross-flow velocity, and by the required trans membrane pressure within the membrane module of the reactor [12]. The optimization of MBRs also requires knowledge of the membrane fouling, the mixing and the biokinetics. MBRs are designed mainly based on biokinetics and membrane fouling considerations, even though the hydrodynamics within an MBR system is of critical importance to the performance of the system. Current design methods cannot directly predict how vessel design (e.g. position of membrane module) affects hydrodynamics, and hence the overall performance.

The modelling of the processes in membrane bioreactors is a relatively new field of research. There are very few examples of modelling such systems where the dynamics of the relationship between the transmembrane pressure, the membrane shear stresses and the flow through the membrane are taken into consideration. However, recently more and more studies are using Computational Fluid Dynamics (CFD) to study and model the flow of fluid through a membrane module as well as to study the factors that cause membrane contamination [12,13]. This is inspired by modelling of other reactors with somewhat comparable patterns such as bubble column reactors [14]. CFD is a technique that can overcome some disadvantages of other methods used in the analysis of membrane modules. It provides a method for predicting the effect of the design features on the hydrodynamics within the MBR. In addition, the effect of the flow regime in the MBR process unit has not been sufficiently understood with respect to MBR design. CFD makes it possible to predict how the geometric characteristics and operating parameters of the membrane reactor will affect its hydrodynamics. This can help the choice of optimal design, and therefore improve the overall performance of membrane bioreactors.

The aim of the proposed study is to develop a CFD model of two and three phase flow (coupling the liquid and gas hydrodynamics) around the submerged membrane module in which the membrane configuration and the aeration conditions are taken into account. Furthermore, some key hydrodynamic factors leading to serious membrane contamination are revealed by studying the effect of the aeration system on the flow velocity, on the membrane surface stress distribution, and by optimizing the MBR.

For this purpose, a computer simulation was carried out to analyse the fluid flow around the surface of the membrane at a different position of the membrane relative to the sparger and to compare the results with those obtained from the simulated hydrodynamic behaviour of a membrane-free bubble column. The results are focused on the operating conditions, the bubble size and the geometry of membrane modules, as well as their impact on the shear stress. As a result of the proposed investigation, evaluation of the effect of membrane localization and air flow sparging on the decrease of membrane fouling is presented. The CFD simulations carried out provide a method for predicting how vessel geometry affects the hydrodynamics, and potentially for optimizing MBR design and performance.

This study develops a CFD model that accounts for aeration and membrane configuration. As expected, aeration was found to be the main mixing mechanism, but CFD was found to be useful in analysing the effect of the geometry / configuration of bioreactors in hydrodynamic conditions. The shear stress distribution on the membrane was also analysed under various operating conditions, such as increased air-flow rate, and a change in the membrane position. CFD simulations were able to provide the effect of design features on the hydrodynamics at a fundamental level, allowing the prediction of the effect of vessel geometry and hydrodynamics on membrane fouling, something of critical importance in the performance of the system.

2. MATERIALS AND METHODS

2.1. Materials

The bubble reactor is modified into a submerged membrane bubble reactor to investigate the effect of the location of the immersed membrane module on the bubble size distribution and on the gas holdup under various operational conditions, such as increasing air flow rate and membrane configuration.

The inclusion of membrane module increased turbulence and increased shear in the upper section of the membrane module. The aeration process tends to be highly turbulent, necessitating a suitable turbulence model. The $k-\epsilon$ turbulence model with enhanced wall treatment was used to properly capture the shear stress at the membrane wall. This turbulence model is commonly used to modeling bubble columns, as it is a tradeoff between computational accuracy and efficiency. To properly capture the shear value at the wall of the membrane, a fine grid was built where the shear stress has an impact.

The complete modeling of the two-phase flow in the ANSYS environment was carried out according to the methodology for geometry, mesh and setup verification, described in previous papers [15,16]. Due to axial symmetry of the object and the operation with volume elements only the 30° sector of the column is simulated, which reduced the calculation time. The main calculation area (the area over the sparger) is meshed by a regular hexagonal dominated mesh (Fig. 2).

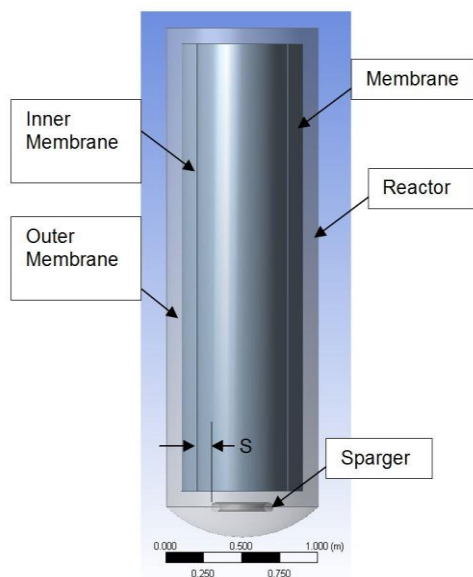


Fig. 1. Geometric model of the bubble column

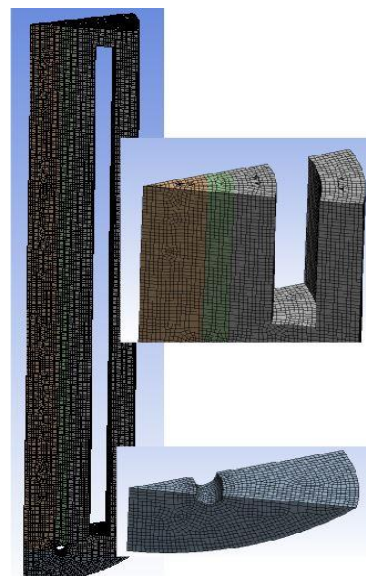


Fig. 2. Mesh

2.2. Methods

The CFD code ANSYS CFX was used to simulate the flow field within the submerged membrane reactor. The numerical simulations were carried out by applying the $k-\epsilon$ model with scalable wall function for turbulence, high resolution advection scheme and second order backward Euler transient scheme for the solution of the Navier-Stokes equation.

When using CFD to model a membrane reactor, the phenomena surrounding the membranes take place on a micro-scale, requiring high mesh resolutions which result in a large number of elements even for the investigation of very small areas. Therefore, an appropriate modeling technique has to be applied to transfer phenomena from the microscale to the macroscale.

The simulation was carried out in a bubble column, which consists of a vertical cylindrical column of height 1.3m and a diameter of 1m, as shown in Fig. 1. The sparger, located at the bottom of the

column, has a toroidal shape with a ring radius of 0.145m and a tube diameter of 0.120m. The submerged membrane is located at different distances from the wall of the column.

The top side of the toroid is perforated with round holes that determine the intensity of the gas flow and the initial diameter of the bubbles. The two-phase liquid systems with air as gas phase and water as liquid phase at the homogeneous regime has been studied computationally.

The simulation was carried out with variations of the following parameters:

- for superficial gas velocity 0,5 m/s and 1,5 m/s;
- for bubble diameter 3mm, 6mm and 12mm;
- for different locations of the membrane module: $S = 0\text{cm}$; 5cm and 10cm.

3. RESULTS AND DISCUSSION

3.1 Effect of aeration

As expected, the aeration was found to be the main mixing mechanism and clearly aeration has a significant influence on the flow field. By increasing the aeration rate, more intense turbulence takes place inside the membrane bubble column. Although the aeration may scour the membrane surface and avoid the formation of cake layer.

Simulation predictions of the distribution of air (water) volume fraction of and of air (water) velocity along the column height are shown in Fig. 3 and Fig. 4. The results obtained enable a successful prediction of the gas holdup and the axial flow velocity of the fluid for the homogeneous regime in the two-phase water-air system.

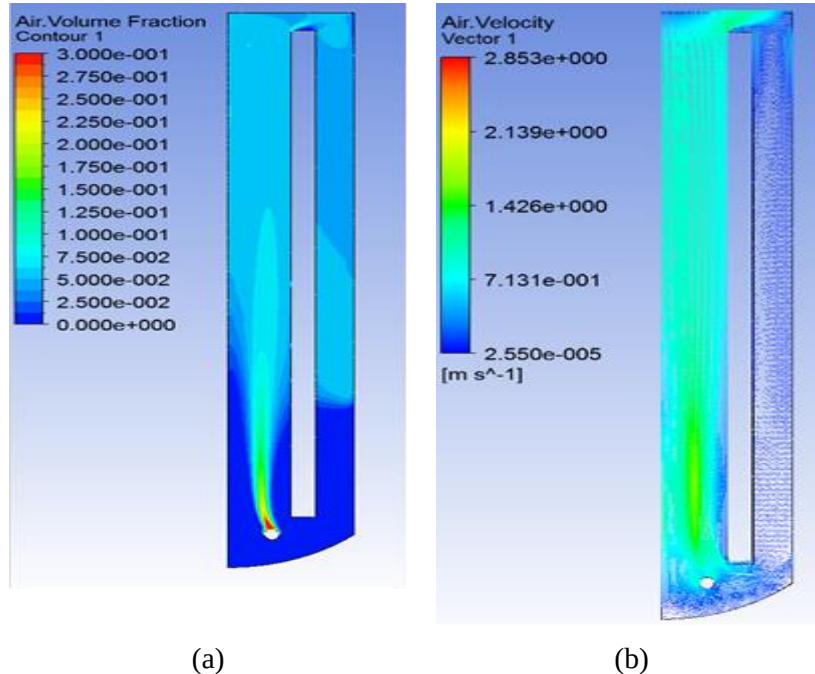


Fig. 3. Gas holdup (a) and air velocity (b) in bubble column at numerical solution with superficial gas velocity 0,5 m/s, bubble diameter 3mm and membrane position $S = 5\text{cm}$

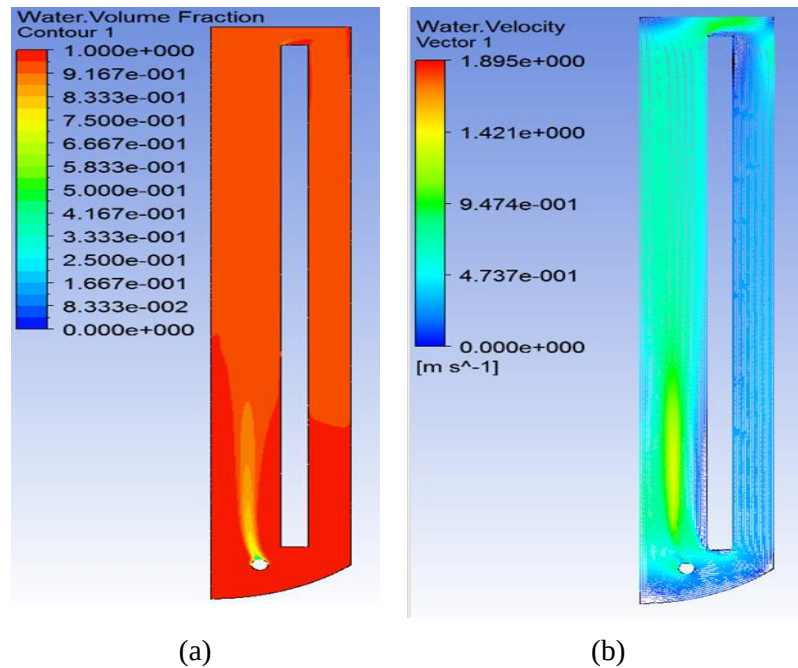


Fig. 4. Water volume fraction in bubble column (a) and water velocity (b) at numerical solution with superficial gas velocity 0,5 m/s, bubble diameter 3mm and membrane position $S = 5\text{cm}$

The velocities below the aerators are generally lower than in the upper zone and increases along the column height. This can be seen in the contour plots of velocity magnitude in Figures 3b and 4b. Higher effective fluid velocities around the membrane also increase the effect of rising bubbles on the shear stresses on the membrane surface.

3.2. Effect of bubble diameter

The graphs of the following figures show that the bubble diameter affects the distribution of shear strain rate along the membrane surface at membrane position $S = 5\text{cm}$ and air superficial velocity 0,5 m/s (Fig. 5) and 1,5 m/s (Fig. 6).

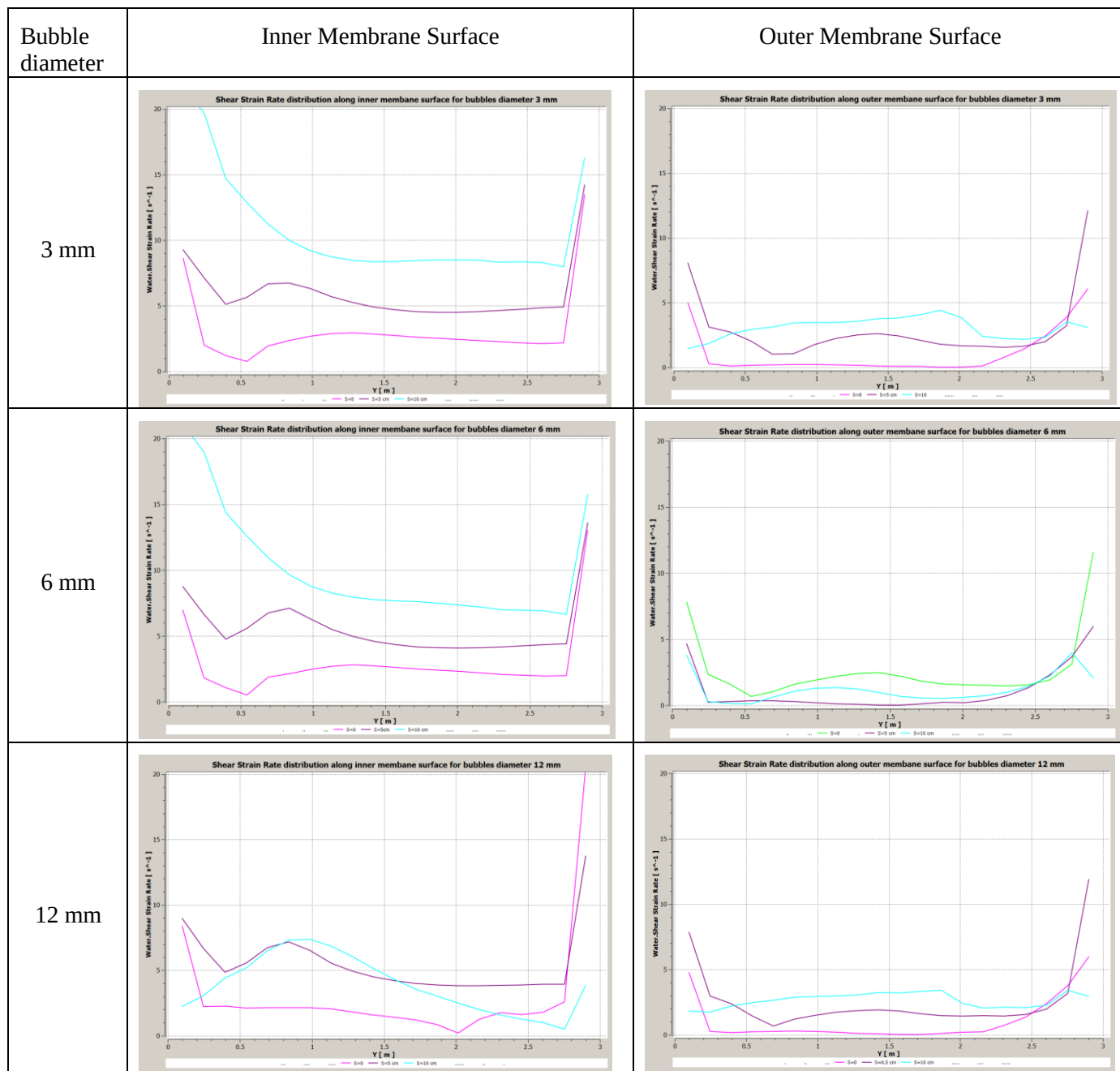


Fig. 5. Distribution of shear strain rate along the inner and outer membrane surfaces at a superficial velocity 0,5 m/s for different bubble diameters: 3mm, 6mm and 12mm respectively

The shear strain rate increases significantly on the upper membrane surface.

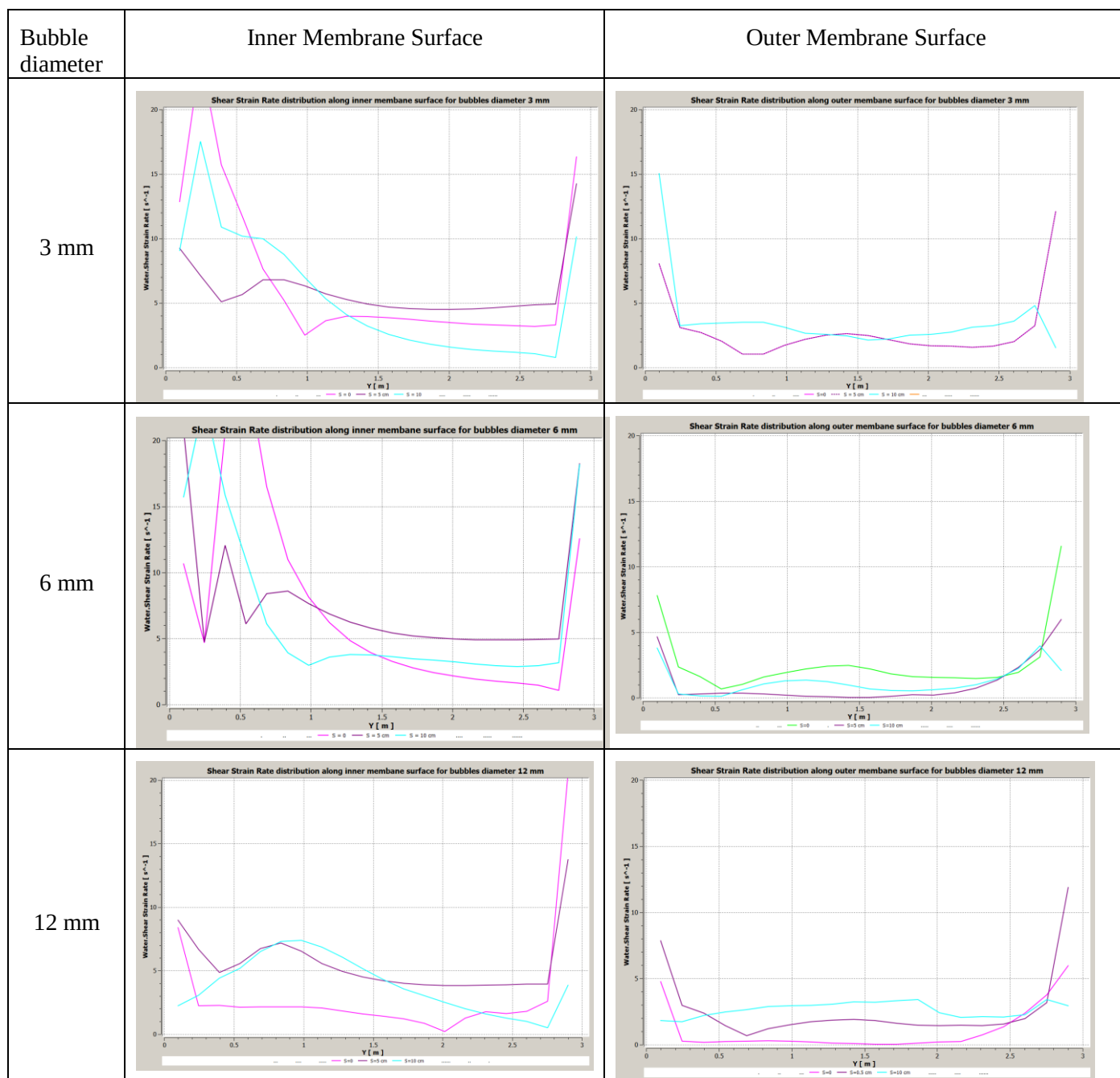


Fig. 6. Contours of shear strain rate produced by air flow with superficial velocity 1,5 m/s on the inner and outer membrane surfaces for various bubble diameters: 3mm, 6mm and 12 mm respectively

Increasing the aeration rate can improve shear-induced diffusion. From figures 5 and 6, it can be seen that the effect of the rising bubbles on the shear strain rate is more significant on the inside of the membrane, therefore the cleaning effect of aeration on the outer surface is negligible.

3.3. Effect of membrane location to the wall of the column

The following graphs show the distribution of shear stress rates at different positions of the membrane: $S = 0\text{mm}$, $S = 5\text{cm}$ and $S = 10\text{cm}$.

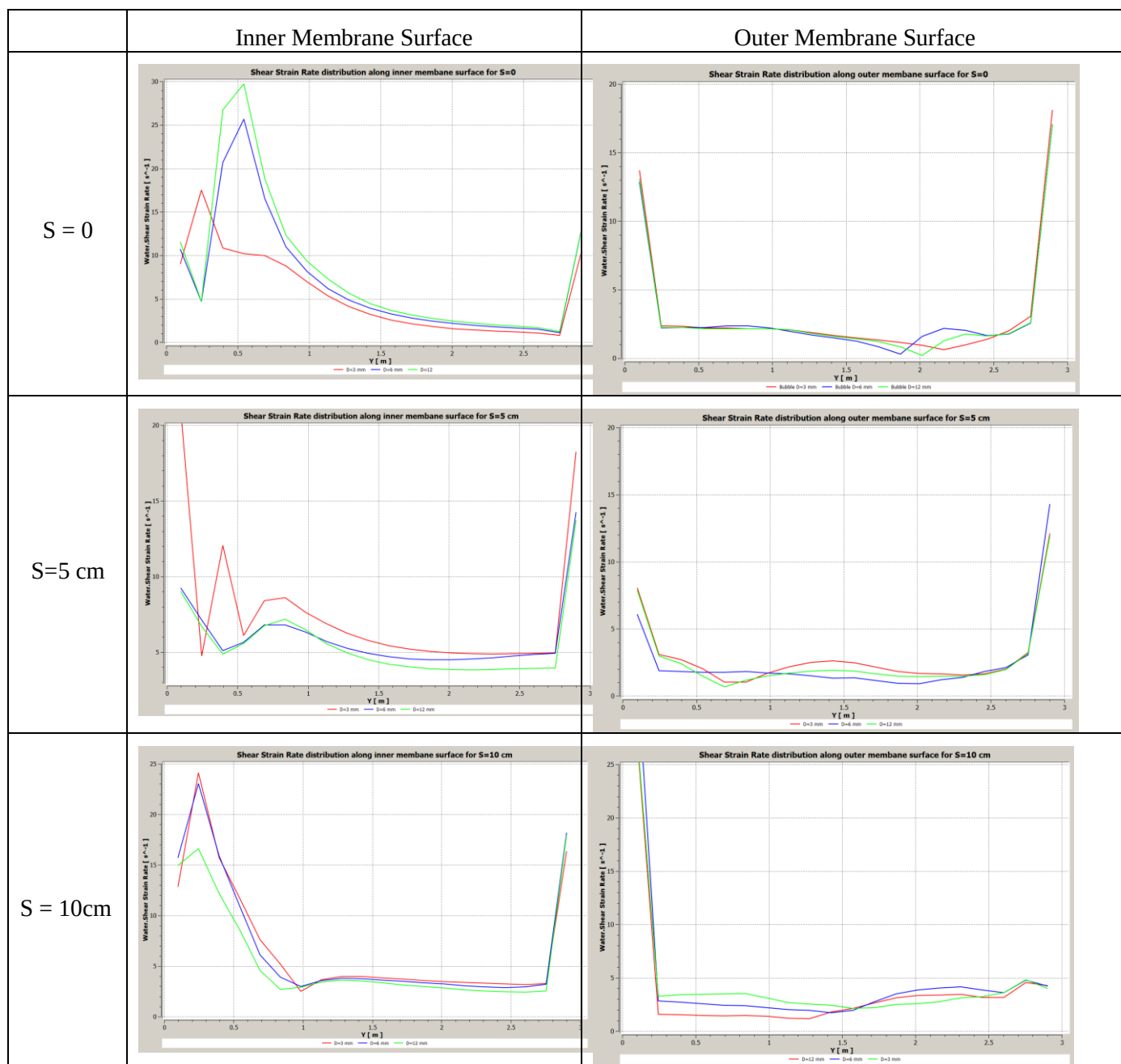


Fig. 7. Contours of shear stress rate produced by air flow on the membrane surface for different membrane locations.

A change of membrane location does not result in any significant improvement on the distribution of shear strain rate. Moreover, it is noticeable that at air velocity 1,5 m/s and membrane location $S = 5 \text{ cm}$, the shear stress rate is greatest and remains constant along the height of the column.

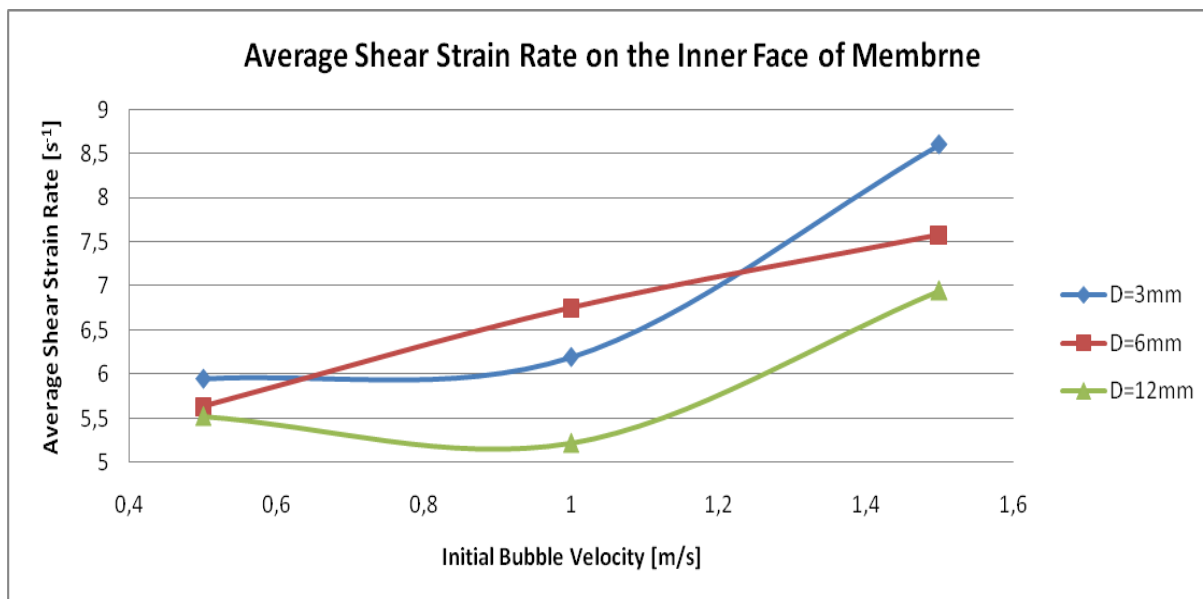


Fig.8. Average shear strain rate distribution depending on the bubble size

Figure 8 shows the influence of the bubble diameter on the average shear strain rate. One can see that shear strain rate grows strongly with increase in bubble size, and that the increase is uniform at bubble diameter 6mm.

The current study on gas sparging focuses on the effect of overall gas flow rates, bubble size, and shear stress rate. It was found that the fluctuation in shear stress affected the flux more than the absolute value of shear stresses induced by the bubbles. The injection of gas increases the turbulence, which reinforces the shear stress. A usual method to reduce the fouling on the membrane is to put a sparger below the membrane surface to inject air near the surface of the membrane, and removing the fouling through the shear stress exerted on the surface.

4. CONCLUSIONS

The CFD model was developed to investigate some key hydrodynamic reasons leading to serious membrane fouling by studying the effect of the aeration system and membrane location on the velocity distribution at the membrane surface, and to optimize the key structure of submerged membrane reactors. Simulation studies were carried to determine the effects of the location of membrane and aeration. They further explained the change in sludge accumulation and its rate by the shear stress profile and concluded that the shear stress is the major hydraulic factor that influences these parameters.

The numerical results of the velocity, volumetric fraction and shear stress rate distributions for the simulations at different geometric characteristics of the membrane module were presented and analyzed. The following trends of flow behavior are registered: (1) shear strain rate dependence on impeller rate is found to be a strong growing function and (2) membrane location from the wall of the column at different distances (from 0cm to 10cm) did not affect the average membrane shear rate significantly. It is the opinion of the authors that the shear stress (generated by the aeration) has a large effect on reduction of filtration resistance in the submerged membrane bubble reactor. Increasing the aeration rate can improve shear-induced diffusion. The best observed shear stress distribution on the membrane surface as obtained by the CFD simulation is at: the location of membrane 5cm, air superficial velocity 1,5 m/s and bubble diameter 3mm.

An increase in superficial air velocity significantly decreases total resistance, since an increase in the air velocity enhances the convective two-phase flow away from the membrane, which subsequently reduces the adsorption of particles on the membrane surface.

The research demonstrates the capability of using CFD to predict how the geometric characteristics and operating parameters will influence the hydrodynamics of the membrane reactor and to optimize the design of membrane module, aeration system and membrane bubble column performance.

ACKNOWLEDGMENTS

The financial support by NSF of Bulgaria, project ДФНИ-ДН 07/11 "Method for assessing the transmission efficiency in integrated processes in a membrane separation bioreactor", Operational Program "Science and education for smart growth" 2014-2020 of the European Union cofounded by the European Social Fund through the project BG05M2OP001-2.009-0015 "Support for the development of capacity of doctoral students and young researchers in the field of engineering, natural and mathematical sciences and RSC of University of Chemical Technology and Metallurgy - Sofia, contract № 11748 "Investigation and optimization of bubble reactor with immersed membrane module using CFD" are gratefully acknowledged.

REFERENCES

1. Yeo A. P. S., Law, A. W. K. & Fane, A. G. Factors affecting the performance of a submerged hollow fiber bundle. *Journal Membr. Sci.* 280 (1), 969–982.
2. Tian J. Y., Chen, Z. L., Liang, H., Li, X., Wang, Z. Z. & Li, G. B. Comparison of biological activated carbon (BAC) and membrane bioreactor (MBR) for pollutants removal in drinking water treatment. *Water Sci. Technol.* 60 (6), 1515–1523.
3. Girar B., Fukumoto, L. R. & Koseoglu, D. S. S. Membrane processing of fruit juices and beverages: a review. *Crit. Rev. Food Sci.* 20 (2), 109–175.
4. Damak K., Ayadi, A., Zeghmami, B. & Schmitz, P. A new Navier-Stokes and Darcy's law combined model for fluid flow in crossflow filtration tubular membranes. *Desalination* 161 (1), 67–77.
5. Liu R., Huang, X., Chen, L., Wen, X. & Qian, Y. Operational performance of a submerged membrane bioreactor for reclamation of bath wastewater. *Process Biochem.* 40 (1), 125–130.
6. Brannock M. W. D., Wever, H. D., Wang, Y., Leslie, G., Chen, V., Leslie, G., Leclech, P. & Li, H. Y. Computational fluid dynamics simulations of MBRs: inside submerged versus outside submerged membranes. *Desalination* 236 (1–3).
7. Ratkovich N., T. Bentzen, M. Rasmussen, Energy consumption in terms of shear stress for two types of membrane bioreactors used for municipal wastewater treatment processes. *Archives of thermodynamics*, vol. 33(2012), 2, 85-106
8. Tsibranska I., S. Vlaev, Tylkowski, B. (2018). The problem of fouling in submerged membrane bioreactors—Model validation and experimental evidence. *Physical Sciences Reviews*, 3(1).
9. Rajabzadeh A. R., Moresoli, C. & Marcos, B. Fouling behavior of electro acidified soy protein extracts during cross flow ultrafiltration using dynamic reversible–irreversible fouling resistances and CFD modeling. *Journal Membr. Sci.* 361 (1–2), 191–205.
10. Zheng J. M., Xu, Z. K., Li, J. M., Wang, S. Y. & Xu, Y. Y. Influence of random arrangement of hollow fiber membranes on shell side mass transfer performance: a novel model prediction. *Journal Membr. Sci.* 236 (1), 145–151

11. Wang X. M., Li, X. Y. & Huang, X. Membrane fouling in a submerged membrane bioreactor (SMBR): characterization of the sludge cake and its high filtration resistance. *Sep. Purif. Technol.* 52 (3), 439–445.
12. Vlaev S., I. Tsibranska, (2016). Shear stress generated by radial flow impellers at bioreactor integrated membranes. *Theoretical Foundations of Chemical Engineering*, 50(6), 959-968.
13. Qing Z., Rongle, X., Xiang, Z. & Yaobo, F. Simulation and optimization of airlift external circulation membrane bioreactor using computational fluid dynamics. *Water Sci.Technol.* 69 (9), 1846–1852.
14. Joshi J.B., Computational flow modelling and design of bubble column reactors, *Chem. Eng. Sci.*, 56 (21–22) (2001) 5893–5933.
15. Stoykova R., D. Moutafchieva, D. Popova, V. Iliev, Gas hold-up prediction in gas-liquid stirred tank reactor using CFD simulation, *Journal of Chemical & Metallurgy*. Vol.49, 5, 2014. p. 469-472
16. Moutafchieva D., V. Iliev, Numerical simulation of bubble breakup and coalescence in bubbling two-phase flow, *Journal of Chem. Techn. and Metallurgy* volume 53, iss.3, 511-517, 2018