



THE CALCULATION AND RESEARCH ON MBR SIMULATION BASED ON CFD

*Xiaohan SUN¹, Chunqing Li¹, Xiangning Chen¹

¹ School of Computer Science and Technology, Tianjin Polytechnic University, Tianjin 300387, China.

*Corresponding Author

ABSTRACT

Membrane bioreactor (MBR) has been successfully applied to biological wastewater treatment, which solves the long-term problem of effective solid-liquid separation and separation. MBR optimization requires knowledge of membrane fouling, mixing, and biomechanics. The MBR is designed primarily for biodynamic and membrane fouling considerations, and fluid dynamics within the system is critical to the performance of the system. Computational fluid dynamics (CFD) provides a method for simulating and predicting MBR. In this paper, the simulation software (Ansys Fluent) was used to establish a membrane bioreactor model with hollow fiber as the membrane component, and the relationship between inlet velocity, height of membrane bioreactor and water yield was studied. The hollow fiber membrane filaments with vertical tangential arrangement were taken as the research object. By optimizing the MBR condition setting, it is concluded that when the inlet velocity and the height of the model are 1:20, the water production effect is the best and the water in the model is the least.

KEYWORDS: CFD; Computational fluid dynamics; MBR; Membrane bioreactor.

1. INTRODUCTION:

Membrane bioreactor (MBR) is a new wastewater treatment process that combines biofilm technology with traditional wastewater treatment technology. MBR has the characteristics of high degree of purification, high degree of equipment integration, and high volumetric load. It has a small footprint, good automation control and low residual sludge production [1]. Brannock M designed a CFD model to explore the effects of ventilation and bio-nutrition on membrane bioreactors [2]. Amini E studied the hydrodynamic properties of shear stress, cross-flow velocity and membrane fouling resistance in membrane bioreactors. Through relevant CFD simulation experiments, the fluid flow, shear stress and cross flow velocity of different mixture concentrations were explored. Experimental results showed that the outermost membrane is more prone to scale formation due to small shear force [3].

Some researches on Computational Fluid Dynamics (CFD) at home and abroad summarize the main factors of membrane fouling formation from membrane fouling factors, operating parameters and membrane materials [4]. Through the numerical simulation to solve the membrane reaction process, this way of solving the problem has attracted the attention of many researchers [5]. The loading density of the flat membrane module was simulated and optimized, and the packing density could be increased by 40% on the original basis [6]. Wang Y studied the hydrodynamic behavior of full-size immersed MBR by using a porous medium model coupled with a three-dimensional multiphase model [7]. By comparing the liquid flow rate and dissolved oxygen with the laboratory-sized immersion MBR reactor, it is concluded that the sludge concentration has a relatively large impact on the experiment [8-9]. The CFD is applied from the fluid mechanics of MBR, and the simulation experiments are carried out by setting relevant parameters to better understand the hydrodynamic behavior of sewage in membrane bioreactor. The computational fluid dynamics method can be used to study the hydraulic parameters of the gas-lift cycle split membrane bioreactor, and related simulation, optimization and sensitivity analysis [10].

In this paper, ANSYS FLUENT was used to complete the simulation experiment. The hollow fiber membrane was used as the membrane material, and the experimental data were referred to uf-4 type membrane components. The porous media model was applied to complete the simulation model calculation.

2. MEMBRANE MODULE MODEL:

The model size is 100*100*100mm (the default arrangement is length*width*height). The cuboid in the center of the model is the water inlet. The size of the water inlet is 10*10*50mm. The two ends of the water inlet are provided with ten hollow fiber tubes perpendicular to the bottom surface and cut respectively, with the size of 0.55*0.55*3.14*50mm. The upper end of the hollow fiber tube is sealed, and the lower end is the outlet. Its inner diameter is 0.6mm and the outer diameter is 1.1mm. The walls are surrounded by the top and bottom. The top and bottom of the model can be opened for easy removal of deposits and membrane cleaning. The three-dimensional rendering of the model is shown in Figure 1.

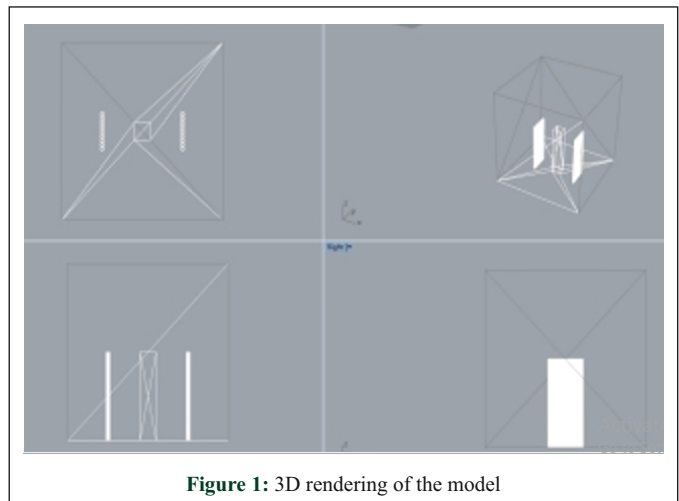


Figure 1: 3D rendering of the model

The membrane module data herein refers to the UOF-4 membrane module, and the parameters thereof are shown in Table 1.

Table 1: Parameters of the model design reference

UOF-4 membrane module	
Single hollow fiber membrane	Inner diameter 0.6mm, Outer diameter 1.1mm
Membrane material	PVDF
Housing material	UPVC
Membrane surface area	40m ²
Maximum inlet pressure	0.3MPa
Maximum transmembrane pressure	0.15MPa
Operating temperature range	5~45°C
pH range	pH 2~10
Pure water initial flux	6000~7000L/h

3. CFD MODELING METHOD:

Basic steps in computational fluid dynamics:

- (1) For the problem under study, establish a relevant mathematical model, and at the same time determine the applied differential equations to describe the relevant hydrodynamic changes;
- (2) use the discretization method to map the continuous variables while making (1) The differential equations are transformed into algebraic equations;
- (3) the discrete positions of (2) are represented by meshing;
- (4) the initial conditions and boundary conditions of the model are determined;
- (5) the control parameters of the model are set;
- (6) Using a computer to solve dis-

crete equations; (7) using graphical software to display calculation results [11-12].

3.1. Porous media model:

The porous media model approach considers the action of the solid structure in the flow region to be the distribution resistance attached to the fluid. The equations and representative parameters are shown in Table 2.

Computational Fluid Mechanics Control Equation:

$$\frac{\partial}{\partial t}(\rho\varphi) + \text{div}(\rho\vec{u}\varphi) = \text{div}(\Gamma\text{grad}\varphi) + S \quad (1)$$

Time item Convection term Diffusion term Generalized source term

Table 2: Formula parameters

Equation	Variable φ	Diffusion coefficient Γ	Source item S
Continuity equation	1	0	0
X-Momentum equation	u	μ	$-\partial p/\partial x + S_{M_x}$
Y-Momentum equation	v	μ	$-\partial p/\partial y + S_{M_y}$
Z-Momentum equation	w	μ	$-\partial p/\partial z + S_{M_z}$
Energy equation	T	k/c	S_T

3.2. Boundary conditions:

Boundary conditions are the most important design parameters for Fluent analysis. The specific types and examples are shown in Table 3 below. At the same time, boundary conditions can be divided into single-sided boundaries and double-sided boundaries. Common single-sided boundaries include axis boundaries, free outflow boundaries and mass flow inlet boundaries. Common double-sided boundaries include fan boundaries, wall boundaries and so on.

Table 3: Types of boundary conditions and examples

Category	Common use model
Import and export boundary conditions	pressure inlet, pressure outlet, speed inlet, air inlet, etc.
Wall condition	wall, symmetry, cycle, axis
Internal unit area	fluid, solid
Internal surface boundary	Radiator, long and short jump, wall, internal interface

4. CFD MODELING PROCESS:

The modeling of the model was completed using the visual drawing software CATIA. The specific effect picture is shown in Figure 2. Then introduce ANSYS ICEM CFD for meshing. Figure 3 is a concrete meshing structure diagram.

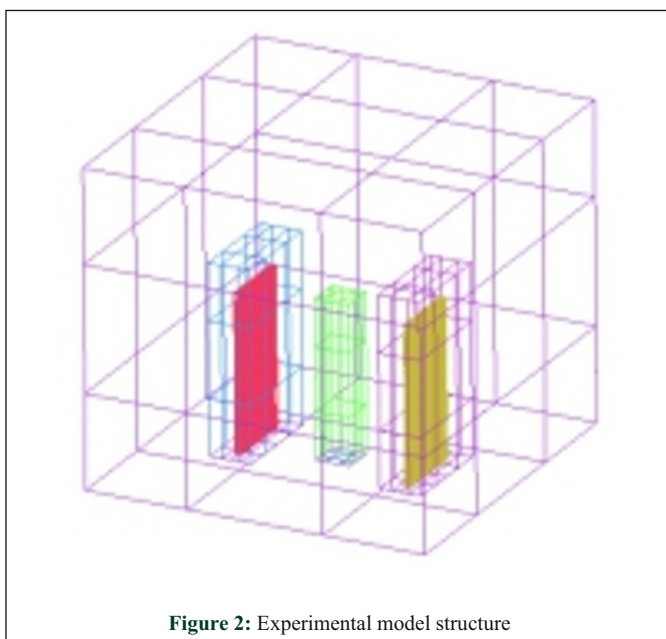


Figure 2: Experimental model structure

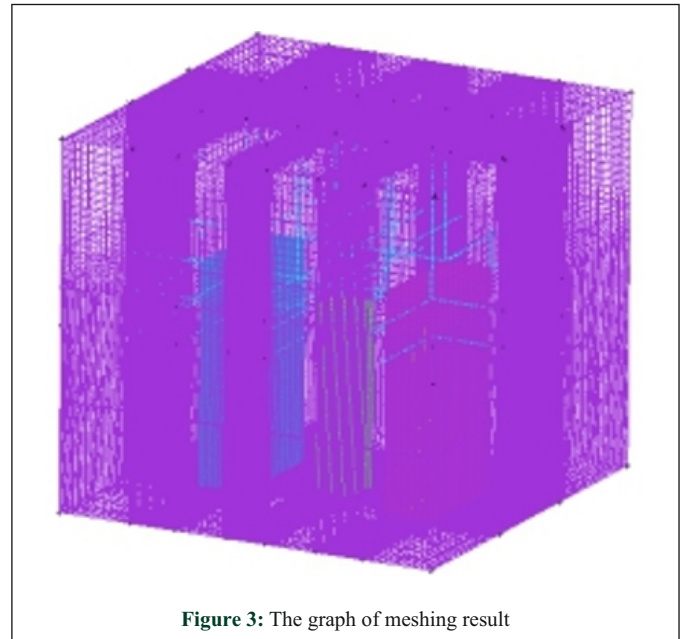


Figure 3: The graph of meshing result

In ICEM CFD, there are two types of meshing: structured meshing and unstructured meshing. Structured meshing usually divides the geometric model into several quadrilaterals or hexahedrons; unstructured meshing usually divides the geometric model into several triangles or tetrahedrons. The specific method of partitioning depends on the actual situation. This experimental model adopts the combination of structural grid and unstructured grid. Twenty tubes were unstructured and applied with a porous media model; others were structured grids. According to relevant experience, this experiment assumes four sets of viscous drag coefficient and inertial drag coefficient for testing. Table 4 shows the viscous resistance and inertial resistance test parameters. Comparing the experimental results, the viscous drag coefficient was $2.111\text{e}+12$ and the inertia drag coefficient was 44000. Then import Fluent, perform the relevant inspection grid, and perform the calculation of the experiment without any errors.

Table 4: Viscous resistance and inertia resistance test parameters

Serial number	Viscous drag coefficient	Inertial drag coefficient
1	$2.111\text{e}+09$	4.4
2	$2.111\text{e}+12$	4.4
3	$2.111\text{e}+12$	440
4	$2.111\text{e}+12$	44000

The number of meshes in this model is nearly 200,000, so the calculation time using Fluent is longer. Through the retrospective method, the influent speed is continuously adjusted. By analyzing the velocity distribution map and the pressure cloud map, the influent velocity is finally 5 mm/s. Under these conditions, the model produces better water production and less water is accumulated inside the model.

The result is shown below. Figure 4 is a flow rate distribution diagram of the experimental model. It can be seen that the flow rate at the top of the model is low, and for the hollow fiber tube portion, the water velocity at the bottom end is large. When the water rushes to the top cover, it falls down and completes the water. Figure 5 shows the distribution of the flow rate map of the model. When the calculation reaches convergence, it can be seen that the model can reasonably complete the filtration when the water intake is stable, and does not accumulate a large amount of water.

Set the number of iterations 500 times to start the calculation. After calculation, the residual curve and the outlet flow are shown in Fig. 6. From the experimental results, we can see that when the iteration is 30 times, the calculation reaches the convergence automatic stop calculation. Figure 7 is the grid quality map. From Figure 7, we can observe that the minimum side length of the block in the model is 0.7, so the mesh quality of the model is high, then save the grid file, prepare the solver for the solution and then save it. The file is ready for post processing.

Fig. 8 is a pressure distribution diagram of a single-row tube. It can be seen from the analysis that the pressure at the upper part of the hollow fiber discharge tube is large due to the position problem of the water inlet, and is sequentially decreased from the top to the bottom. The bottom of the tube is caused as a water outlet and the pressure is small.

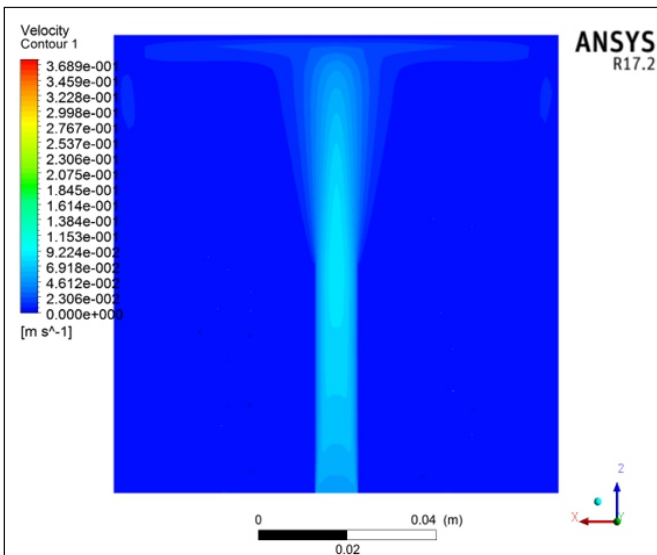


Figure 4: Inlet flow velocity distribution map

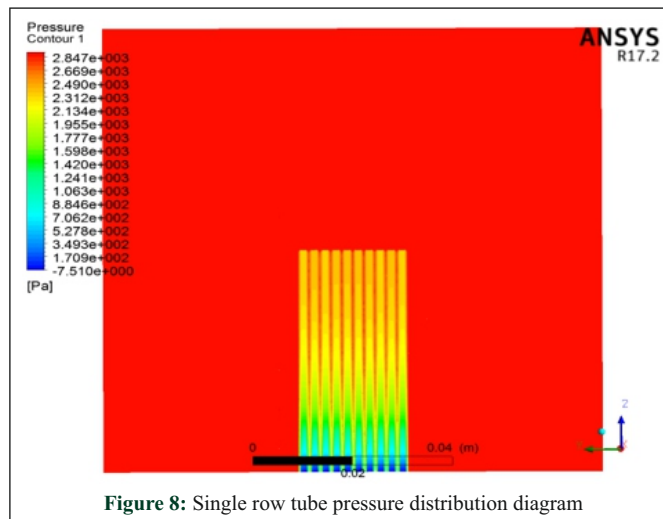


Figure 8: Single row tube pressure distribution diagram

5. CONCLUSION:

In this chapter, based on CFD, the filtration process of hollow fiber-based membrane modules was studied, and the velocity distribution, pressure distribution and flow velocity distribution of hollow fiber membrane filaments were analyzed. The wall of the simulation model uses a porous media model to bring the model closer to the actual hollow fiber membrane filament. The pressure cloud and streamline diagram of the post-processor directly describe the operation of the MBR membrane module, allowing us to clearly observe the flow and stress of the wastewater in the membrane module. The simulation experiment continuously adjusts the inlet speed and judges the water production effect from the model calculation results. When the water inlet speed and the model height are 1:20, the water production effect is the best, and the model has the least water. Modeling and simulation of MBR system using computational fluid dynamics is a new research idea and method. Applying CFD model to MBR project can save a lot of MBR engineering design and engineering implementation cost, and it has certain research on future MBR field. Theoretical and practical significance.

Acknowledgments:

This work was supported in part by the National Natural Science Foundation of China (Grant no. 51378350), the National Natural Science Foundation of China (Grant no. 50808130) and the National Natural Science Foundation of China (Grant no. 21506159).

REFERENCES:

1. Naghizadeh A, Mahvi A H, Mesdaghinia A R, et al. Application of MBR Technology in Municipal Wastewater Treatment[J]. Arabian Journal for Science and Engineering, 2011, 36(1):3-10.
2. Brannock M, Leslie G, Wang Y, et al. Optimising mixing and nutrient removal in membrane bioreactors: CFD modelling and experimental validation.[J]. Desalination, 2010, 250(2):815-818.
3. Amini E, Mehrnia M R, Mousavi S M, et al. Experimental Study and Computational Fluid Dynamics Simulation of a Full-Scale Membrane Bioreactor for Municipal Wastewater Treatment Application[J]. Industrial & Engineering Chemistry Research, 2013, 52(29):9930-9939.
4. Wang Jie, Wu Yi, Luo Nan, et al. Simulation of internal flow field and gas distribution in membrane bioreactor by CFD method[J]. Civil Engineering and Environmental Engineering, 2014, 36(02): 119-125.
5. ZHANG Qing, FAN Yao-bo, WEI Yuan-shou, et al. CFD Simulation and Optimization of Airlift Cycle Split MBR[J]. Membrane Science and Technology, 2013, 33(04): 107-119.
6. Zhang Yu, Wu Zhichao, Zang Lili, et al. Research on CFD design and operation optimization of flat membrane MBR[J]. Journal of Environmental Engineering, 2016, 10(02):735-741.
7. Wang Y, Brannock M, Cox S, et al. CFD simulations of membrane filtration zone in a submerged hollow fibre membrane bioreactor using a porous media approach[J]. Journal of Membrane Science, 2010, 363(1-2):57-66.
8. Yang M, Yu D, Liu M, et al. Optimization of MBR hydrodynamics for cake layer fouling control through CFD simulation and RSM design.[J]. Bioresource Technology, 2016, 227:102.
9. Zhu Kunpeng. CFD design and optimization of integrated MBR ship domestic sewage treatment equipment [D]. Dalian: Dalian Maritime University, 2017.
10. Wang Fujun. Computational Fluid Dynamics Analysis [M]. Beijing: Tsinghua University Press, 2004.
11. Chai Jintao. Study on the effect of membrane material on ultrafiltration of secondary effluent from municipal wastewater [D]. Xi'an: Xi'an University of Architecture and Technology, 2006.
12. ZHAO Ying, YU Dan-dan, QIN Dong-ping, et al. Effects of PAC dosage on the properties of MBR mixture and membrane fouling[J]. Water Treatment Technology, 2005(11): 52-55.

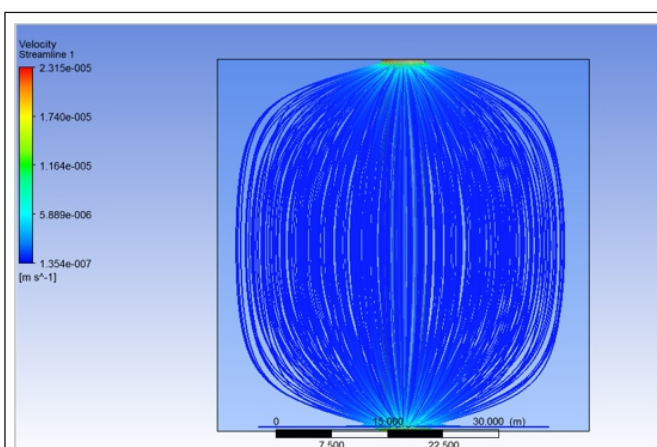


Figure 5: Model flow rate distribution map

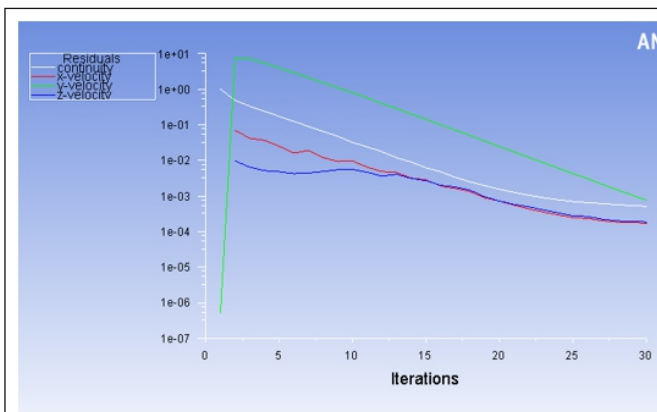


Figure 6: Model flow rate distribution map

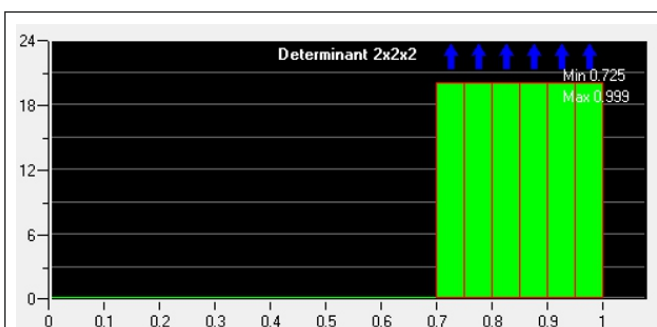


Figure 7: Grid quality map