IOP Conference Series: Earth and Environmental Science

PAPER • OPEN ACCESS

Establishment of CFD model for rotating cross-flow membrane separation device

To cite this article: Zhihao Chen *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **233** 052053

Establishment of CFD model for rotating cross-flow membrane separation device

Zhihao Chen[1a], Liuhuan Li[1b], Dazhou Cheng[1c]

[1] College of Ocean Science and Engineering, Shanghai Maritime University, China

[a]954707198@qq.com, [b]825622755@qq.com, [c]2253393907@qq.com

Abstract: An initial CFD model which was based on an active rotary cross-flow membrane separation device was established. Fluent commercial software was used to carry out numerical simulation of the two axis membrane disk device. It was found that as the inflow velocity increased, the particle residence time decreased and gradually became stable. Therefore, controlling the inflow velocity can improve the load of diaphragm and the efficiency of purification.

1. Introduction

At present, among many water treatment technologies, membrane treatment technology is considered to be the most promising water purification technology[1]. The traditional static filtration technology will rapidly generate the accumulation of particles on the surface of the membrane during filtration, which will affect the efficiency of filtration[2]. Active rotary cross-flow filtration technology can change this situation. It has the advantages of reducing membrane fouling and increasing membrane flux.

2. Establishment of CFD model

Computational Fluid Dynamics (CFD) techniques are used to perform fluid dynamics simulations of membrane processes for Rotary Cross-Flow Membrane Separation (RCM).

The fluid flow conforms to the basic fluid mechanics equation, the continuity equation and the momentum conservation equation. When the liquid is mixed with solid or gas, the momentum and energy equations are complex. But as a whole, it follows the basic conservation equation. Most of the liquid in this study is in turbulent state, so we must also consider the turbulent energy dissipation equation.

2.1. Governing equations for computational fluid dynamics simulation

Navier-Stokes equation is the equation derived from the continuity equation and momentum conservation equation. The equation is not closed. It is a motion equation which describes the momentum conservation of viscous incompressible fluids. Its vector form is:

$$\rho \frac{dv}{df} = -VP + \rho f + \mu \Delta v$$

The above formula is the basic form of the N-S equation. When solving with Fluent software, the N-S equation is solved discretely in combination with other diffusion terms, turbulent energy generation terms, dissipative terms and boundary layer conditions, and the final simulation results are obtained.

2.2. Turbulence model for CFD simulation

For any fluid flow, no matter how complex its own state, must satisfy the continuity equation and N-S equation. The N-S equation is a partial differential equation with nonlinear characteristics. It is impossible to obtain analytical solution accurately at present. Therefore, using discrete numerical methods is the main way to solve the N-S equation[3].

The eddy viscosity models commonly used in the RANS model are: single equation model(k-equation model), zero equation model, double equation model(k-ε model, k-ω model), etc. The k-ε model is chosen in the CFD simulation.

The k-ε model was first proposed by Wilcox. It belongs to the eddy viscosity simulation of two equations. It has a good application to the simulation of the wall of rotating flow[4]. Its equation is as follows:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho k u_j)=\frac{\partial}{\partial x_j}((\mu+\sigma^*\mu_t)\frac{\partial k}{\partial x_j})+P_k-\beta\cdot\rho\omega k$$

$$\frac{\partial\rho\omega}{\partial t}+\frac{\partial}{\partial x_j}(\rho\omega u_j)=\frac{\partial}{\partial x_j}((\mu+\sigma\mu_t)\frac{\partial k}{\partial x_j})+\alpha\frac{\omega}{k}P_k-\beta\rho\omega^2$$

In the above formula, k is the turbulent kinetic energy, ω is the dissipation rate, and the three items on the right are the diffusion term, the turbulent energy generation term, and the turbulent energy dissipation term. Among them, the turbulent energy generation term P_k is:

$$P_k=-\rho\overline{u_i'u_j'}\frac{\partial u_i}{\partial x_j}$$

Eddy viscosity is defined as:

$$\mu_t=pk/\omega$$

The values of α, β and other parameters in the above formula can be referred to the study of k-ω two equations[5].

2.3. Device and simulation

The active cross-flow filtration device of this experiment is a self-made active cross-flow flat membrane separation device, and the membrane module size (length, width and height) used is 120 cm * 40 cm * 40 cm.

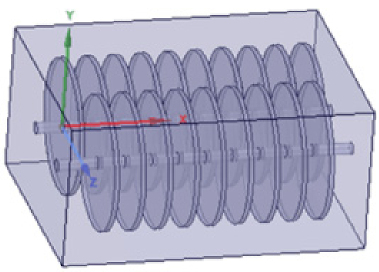

Fig.1.simplified diagram of device structure

The raw water enters the device at a different speed in the positive direction of the y-axis. After being filtered by the membrane module, the raw water is derived from two collecting pipes with diaphragms along the x-axis. In previous experiments, the measured membrane flux was $100cm^3/(m^2*h)$ and the device speed was 130rpm. During this simulation, the velocity of the water flows into the device by 0.8m/s, 1m/s and 1.2m/s respectively.

The physical model of the device is first established using the DM in the Workbench to create a three-dimensional physical model. The characteristics of the MRF model used in this study need to be considered when building the model. The rotation of the rotating phase is used to simulate the rotation of the disk.

2.4. Simulation method of the device

After the 3D model is built, it needs to be meshed. Use the Meshing in the whole Fluent solution of WorkBench to divide the grid. Accuracy can be achieved by using Meshing.

In Meshing, we first set the mesh quality to Fine and set the correlation coefficient to 100 to ensure the quality and accuracy of the mesh. The whole division is based on a tetrahedral mesh, with a total of $4 * 10^5$ units. After meshing and setting, import Fluent to set specific conditions. Define the material of the MRF as a liquid material, to eliminate the warning that the solid boundary is too close to the liquid boundary when the mesh is imported. In the model (Model), it is set to the standard k-ε model.

Regarding the problem of calculation accuracy, there is a difference between first-order and multi-order. Although the first-order precision is not high in accuracy, the degree of convergence is good. So the first-order precision is used first in the calculation. If the convergence of the residual curve is good, it can be returned to improve the accuracy of these items one by one.

Since the initial setting is constant current (MRF is generally only used for constant current calculation), we only need to calculate a fixed number of steps. For the accuracy of the calculation, set the number of calculation steps to 4000.Then start the calculation.

3. Numerical Simulation

3.1. Simulation results of stable operation

After simulation calculation, a stable operation phase with an inflow velocity of 1 m/s is obtained.The Fig.2 below is the axial velocity vector profile. The direction of the arrow in the graph represents the direction of fluid flow, and the size of the arrow represents the size of the flow. From the local magnification, it can be seen that the velocity distribution near the film is basically stable, and there is a circulating flow near the wall. This circulation flow can improve the cross-flow effect of the fluid to a certain extent, and relieve the deposition of particles on the surface of the membrane.

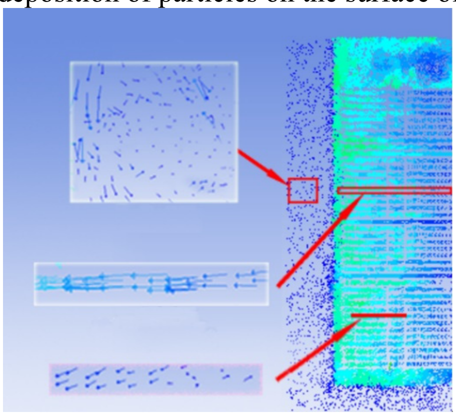

Fig.2. velocity vector of axial membrane tray

3.2. Comparison of particle residence time under different inflow velocity conditions

In order to study the residence time of particles under different inflow conditions, three different inflow velocities of 0.8m/s, 1m/s and 1.2m/s are selected. The Fig.3 below is a graph of the residence time of the particles at different inflow velocities of 0.8 m/s, 1 m/s, and 1.2 m/s.

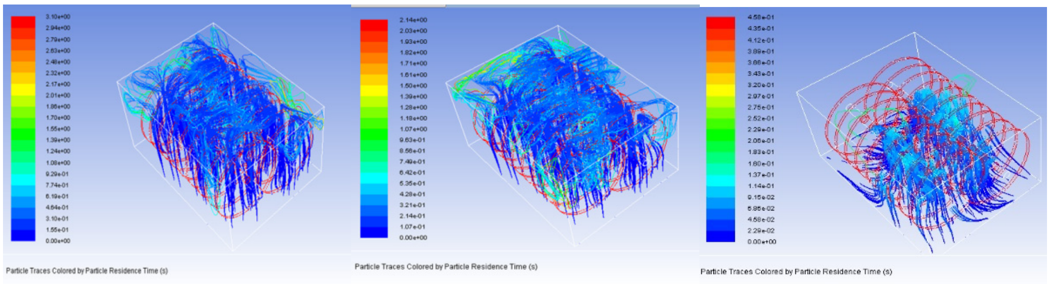

Fig.3. Particle residence time diagram at inflow velocity of 0.8m/s, 1 m/s and 1.2 m/s

Take the disk diameter direction as the abscissa and the dwell time as the ordinate. The midpoint of the staggered area is the origin, and the intermediate diaphragm interlaced area is 8cm. A function image of residence time with disk diameter is made.

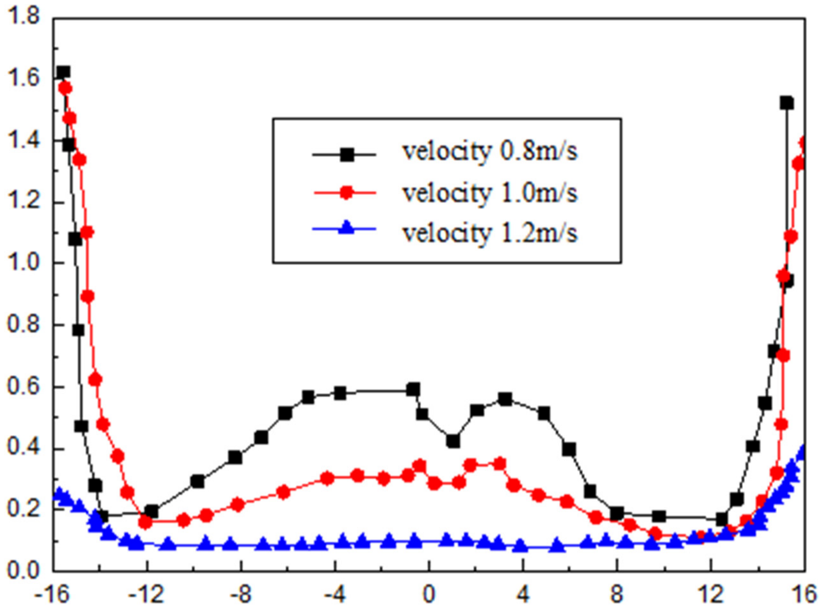

Fig.4. Particle residence time diagram under different inflow velocities

It can be seen from Fig.4 that as the velocity increases, the particle residence time decreases and gradually becomes stable, and the peak value becomes less obvious, and the difference in particle residence time gradually becomes smaller in the entire advice. It can be understood that as the inflow velocity increases, the sludge particles are gradually released from the hydraulic dead zone, causing the particles to continuously circulate as the device operates. When the inflow speed is not large, the particles are easily locked in the hydraulic dead zone.

4. Conclusions

It can be understood that the basic characteristics of the flow field in the active cross-flow filtration device, especially the distribution of fluid characteristics between the membrane discs. It is proved that the model is feasible for simulating the rotating cross-flow membrane separation device.

It is found in simulation that as the velocity increases, the particle residence time decreases and gradually becomes stable, and the peak value becomes less obvious. Since the particles are easily affected by the inflow velocity, controlling the inflow velocity can better improve the load on the diaphragm.

References

[1] Bing Y, Guisheng F, Zhenhai L. Comparison between membrane water treatment technology and traditional methods [J]. Journal of Taiyuan University of Technology, 2004, (02):155-

159+163
[2] Weiqing K, Lijie S, Youjing Z, et all. Application of Membrane Separation Technology in Water Treatment [J]. Inorganic salt industry, 2014,46(05):6-9
[3] Ziniu W. Fundamentals of computational fluid dynamics [M]. Beijing: Science Press
[4] Wilcox, D.C. Reassessment of the Scale-Determining Equation for Advance Turbulence Models [J].Aiaa Journal,1988,23(11),1299-1310.
[5] Menter F R. Two-equation eddy-viscosity turbulence models for engineering applications [J]. AIAAJ, 1994, 32(8):1598-1605.
[6] Wu Y. Study on k-ω two-equation turbulence model [D], Beijing: Beihang University