Contents lists available at ScienceDirect

Chemical Engineering Research and Design

journal homepage: www.elsevier.com/locate/cherd

Effect of the baffle design and orientation on the efficiency of a membrane tube

Houari Ameur[a,*], *Djamel Sahel*[b]

[a] *Department of Technology, Institute of Science and Technology, Ahmed Salhi University Center of Naâma (Ctr Univ Naâma), PB 66, 45000, Algeria*
[b] *Laboratory of Gaseous Fuels and Environment, Faculty of Mechanical Engineering, University of Science and Technology USTO-MB, PB 1505, El-M'Naouar, Oran 31000, Algeria*

ARTICLE INFO

Article history:
Received 19 April 2016
Received in revised form 22 October 2016
Accepted 6 November 2016
Available online 14 November 2016

Keywords:
Membrane tube
Baffle design
Hemispherical baffle
Filtration
Turbulent flows

ABSTRACT

A numerical investigation is carried out to examine the effect of a new baffle design in a membrane tube on the hydrodynamics and filtration efficiency. Two different orientations of hemispherical baffles named as RO baffle for the Right Orientation and LO baffle for the Left Orientation, respectively, are explored. Two values of the carbonate calcium suspensions are used: 5 and 10 g/L. The axial velocity, stream function, static pressure, wall shear stresses and turbulent characteristics are the physical parameters utilized to evaluate the filtration performance. The obtained results showed that the presence of an array of hemispherical baffles can develop the local shear stresses on the membrane surface and create the fluid eddy movement which enhances considerably the filtration performance. When the feed concentration is 5 g/L and in a comparison with the unbaffled tubes, the RO and LO cases achieved an increase in the filtration flux rate by 57% and 64%, respectively. For the second feed concentration (10 g/L), the enhancements are 85% and 96% for the RO and LO cases, respectively. In a comparison between the LO and RO cases, the LO baffle gives the best performance. Our results were compared with experimental data and a satisfactory agreement has been found.

© 2016 Institution of Chemical Engineers. Published by Elsevier B.V. All rights reserved.

1. Introduction

The membrane tubes are widely used in several industrial processes, such as water treatment, removal of heavy metal, desalination and other industrial fields. However, the major problem in using the membrane tube in these processes is the concentration polarization, which reduces the filtration flux due to the formation of the hydrodynamic boundary layer, resulting then in the formation of a gel on the membrane surface. Baffles or turbulence developers inside the membrane tube are considered as a very successful technique to enhance the filtration flux. These geometrical configurations help to stop the appearance of the boundary layer and modify the flow structures on the membrane surface, which improves the filtration flux phenomenon. Numerous authors proposed various designs for developing the filtration execution in membrane tubes (Chen et al., 2014; Chiu and James, 2006; Ghaffour et al., 2004; Krstic et al., 2006; McDonough et al., 2015; Pal et al., 2008). The majority of these works are based on physical parameters related to the flow structure such as the velocity, wall shear stress, turbulence kinetic energy, dissipation energy and static pressure.

Some researchers have interested to the study of mass transfer in a baffled membrane system by means of experiments (Brunold et al., 1989; Finnigan and Howell, 1989; Mackley et al., 1990; Wang et al., 2013). However, and according to the development of data-processing tools and numerical methods during the last decades, the computational fluid dynamic (CFD) is becoming a very suitable device in the membrane science field, where satisfactory results may be obtained with less time and expense of energy (Chen et al., 2013; Liu et al., 2015; Wang et al., 1994). Liu et al. (2009) have studied numerically the effect of central and wall baffles on the performance of a membrane tube. Their results showed that the presence of an array of baffles inside the membrane

* *Corresponding author.*
E-mail address: houari_ameur@yahoo.fr (H. Ameur).
http://dx.doi.org/10.1016/j.cherd.2016.11.005
0263-8762/© 2016 Institution of Chemical Engineers. Published by Elsevier B.V. All rights reserved.

Nomenclature

b_t	Baffle thickness (m)
d	Baffle diameter (m)
D	Tube diameter (m)
E_s	Eddy size (m)
L	Tube length (m)
l	Distance between two consecutive baffles (m)
P	Pressure (Pa)
t	Time (s)
u	Velocity (m/s)
u_i	Axial velocity (m/s)
u_j	Radial velocity (m/s)
Greek letters	
ρ	Density (kg/m^3)
μ	Dynamic viscosity (Pa s)
δ_{ij}	Kronecker sign
u_t	Turbulent viscosity (Pa s)

tube cause a frequent change of the flow direction, where the central baffle presents higher shear stresses on the tube wall. Consequently, the central baffle is the best choice for the filtration execution. In the same framework, Ahmed et al. (2011) and Monfared et al. (2012) confirm that the presence of baffles inside the membrane tubes increases the wall shear stress, which enhances the filtration performances. Rainer et al. (2002) simulated the filtration phenomenon in the presence of rotating discs inside the tubes and reported that the use of bent scrapers yields significant developments in the passing flow and turbulence behavior in the zone behind scrapers. Jafarkhani et al. (2012) explored the effects of baffle orientation angles (90° and 180°) and diameter ratio (1–3) on the filtration flux. They found that the further extension of the baffle orientation angles from 90° to 180° enhances considerably the filtration by increasing the fluid average velocity, shear stress and mass transfer on the tube wall.

In this paper, turbulent flows in membrane tubes equipped with an array of hemispherical baffles are analyzed with the help of a CFD method. We focus on effects of the baffle orientation on the efficiency of such systems.

2. Mathematical and numerical solution

2.1. *Problem geometry*

The problem geometry concerns a horizontal tube with inner diameter (D) of 15 mm and length (L) of 200 mm, as illustrated in Fig. 1. Two different orientations of hemispherical baffles are realized, which are: LO baffle (Case A) for the Left-Orientation and RO baffle (Case B) for the Right-Orientation, respectively. The baffle thickness (b_t) is 1 mm and its diameter (d) is 12 mm. The distance between two consecutive baffles (l) is equal to 22.5 mm. The first baffle is placed at a distance of 22 mm from the inlet tube.

Two methods are suggested for the installation of baffles in the tube: baffles may be mounted on a shaft passing through the center of each baffle, or each baffle may be inserted on small supports fixed on the wall of the membrane tube.

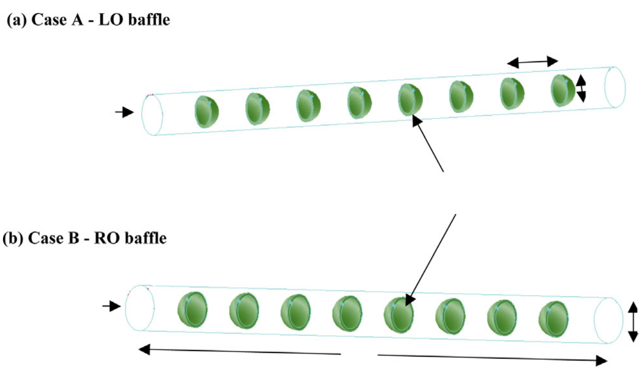

Fig. 1 – Baffle configurations.

2.2. *Governing equations*

The numerical model of Newtonian, incompressible and isothermal fluid, with constant physical properties (water) in a cylindrical tube is described by the continuity and Reynolds averaged Navier–Stokes equations, as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \left(\frac{2}{3}\mu\frac{\partial u_i}{\partial x_i}\right)\right] - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \tag{2}$$

The Reynolds-averaged approach to turbulence modeling requires the modeling of the Reynolds stresses $-\rho\overline{u_i' u_j'}$ in Eq. (2). The Boussinesq hypothesis relates the Reynolds stresses to the mean velocity gradients as seen in the equation below:

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_i}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t\frac{\partial u_i}{\partial x_i}\right)\delta_{ij} \tag{3}$$

where the turbulent viscosity (μ_t) is described by:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}, \quad C_\mu = 0.085 \tag{4}$$

The Reynolds number is defined in the hydraulic diameter as:

$$Re = \rho U_m D_h/\mu \tag{5}$$

For closure of the equations, the *RNG* k-ε model was used. The *RNG* k-ε turbulence model is derived from the instantaneous Navier–Stokes equations, using a mathematical technique called "renormalization group" (*RNG*) methods. This model is very adequate for the prediction of turbulent flows in the membrane tubes (Ahmed et al., 2011; Liu et al., 2009). The turbulent kinetic energy (k) and turbulent dissipation rate (ε) are determined by the following equations:

$$\frac{\partial \rho}{\partial t}(\rho k) + \frac{\partial \rho}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon \tag{5}$$

$$\frac{\partial \rho}{\partial t}(\rho\varepsilon) + \frac{\partial \rho}{\partial x_i}(\rho\varepsilon\upsilon_\iota) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_\tau}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{\kappa}\Pi_\kappa - C_{2\varepsilon}^*\rho\frac{\varepsilon^2}{k} \tag{6}$$

Download English Version:

https://daneshyari.com/en/article/4987397

Download Persian Version:

https://daneshyari.com/article/4987397

Daneshyari.com