Computers and Chemical Engineering 61 (2014) 30–37

Contents lists available at ScienceDirect

Computers and Chemical Engineering

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

Numerical study on hydrodynamic characteristics of plate-fin heat exchanger using porous media approach

Weiping Wang[a], Jinxing Guo[a], Shuwen Zhang[b], Jian Yang[a,*], Xianting Ding[c], Xuehua Zhan[b]

[a] Institute of Process Equipment, Zhejiang University, Hangzhou 310027, China
[b] Hangzhou Hangyang Co., Ltd, Hangzhou 310014, China
[c] School of Biomedical Engineering, Med-X Research Institute, Shanghai Jiao Tong University, Shanghai 200240, China

ARTICLE INFO

Article history:
Received 13 June 2013
Received in revised form 10 September 2013
Accepted 16 October 2013
Available online 24 October 2013

Keywords:
Computational fluid dynamics
Flow distribution
Perforated fins
Plate-fin heat exchanger
Porous media

ABSTRACT

A numerical plate-fin heat exchanger (PFHE) model was proposed to investigate the hydrodynamic characteristics of a full-size PFHE by using the porous media approach. Based on the model, effects of the fluid dynamic viscosity and perforated fins on flow distribution and pressure drop of the PFHE were studied. The results showed that flow distribution of the PFHE was improved by increasing the fluid dynamic viscosity or adding perforated fins in each fin channel, but at the cost of an increased pressure drop. Therefore, the relationship between flow distribution and pressure drop was further analyzed under various Reynolds numbers. Based on the results, a correlation among flow distribution, pressure drop, and Reynolds number was derived. Finally, two strategies, the fin channel-based strategy and the header-based strategy were proposed and numerically verified to improve flow distribution of the PFHE. Our results indicate that the first strategy is better than the latter one.

1. Introduction

Plate-fin heat exchanger (PFHE) known for their compactness and low weight is broadly applied in a wide range of chemical engineering applications. A typical PFHE is commonly made up of layers of corrugated sheets, which are separated by flat plates to create a series of finned chambers. The design of a PFHE is a complex task with special requirements to be considered, such as temperature difference, flow distribution, and pressure drop. Among these factors, flow distribution and pressure drop of the PFHE may have a significant impact on the thermal and mechanical performance of the PFHE. Thus, it is necessary to operate the PFHE with good hydrodynamic characteristics.

Over the last few decades, many theoretical researches have determined the relationship between thermal performance and flow nonuniformity of the PFHE. Using a successive substitution technique, Chiou (1978) presented a mathematical method to determine the heat exchanger effectiveness by considering the effect of flow maldistribution. Müller-Menzel and Hecht (1995) theoretically discussed the occurrence of various flow patterns in a PFHE and their impact on the overall performance of the heat exchanger. The combined effect of longitudinal heat conduction and flow nonuniformity on a crossflow PFHE based on the finite-element method was analyzed by Ranganayakulu and Seetharamu (2000). They found that the performance deteriorations are quite significant in some applications.

In recent years, some efforts have been taken to investigate the hydrodynamic characteristics in such complex geometry by using experimental methods. Jiao, Zhang, and Jeong (2003) and Jiao and Baek (2005) conducted an experimental investigation into the effects of the distributor and header configuration on flow distribution in the PFHE. The studies proved that flow distribution in the PFHE can be effectively improved by optimum design of the distributor and header configuration. By using particle image velocimetry (PIV), Wen, Li, Wang, and Zhou (2006) investigated the turbulent flow structure inside the entrance of a PFHE, streamline and velocity contour graphs of different cross-sections were achieved for three distinct header configurations. Since computational fluid dynamics (CFD) simulation can provide an effective platform where various optimal designs can be tested and determined at a relatively low cost, more recently, several reports have studied the hydrodynamic characteristics of the PFHE by applying CFD technique. Zhang and Li (2003), Wen, Li, Zhou, Zhang, and Wang (2004), Wasewar, Hargunani, Atluri, and Kumar (2007)

* Corresponding author at: Yuquan Campus, Zhejiang University, No. 38 Zheda Road, Hangzhou 310027, Zhejiang, China. Tel.: +86 571 8795 3098; fax: +86 571 8795 3393.
E-mail address: zdhjkz@zju.edu.cn (J. Yang).

0098-1354/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compchemeng.2013.10.010

$C_{1\varepsilon}$	constant for ε
$C_{2\varepsilon}$	constant for ε
C_1	viscous resistance factor (m^{-2})
C_2	inertial resistance factor (m^{-1})
D_h	hydraulic diameter (mm)
d	diameter (mm)
f	friction factor
G_k	generation of k
h	fin height (mm)
k	turbulence kinetic energy
L	fin length (mm)
m	mass flow rate (kg/s)
n_r	row number of holes
n	number of fin channels
P	Pressure (Pa)
r	radius of holes (mm)
Re	Reynolds number
STD	standard deviation (%)
S	source term
s	fin spacing (mm)
t	fin thickness (mm)
U	velocity vector
v	fluid velocity (m/s)
Greek symbols	
α	permeability
Γ	diffusion coefficient
Δn	porous media thickness (mm)
ΔP	pressure drop (Pa)
ε	turbulence dissipation function
μ	dynamic viscosity (Pa s)
ρ	density (kg/m^3)
σ	porosity (%)
σ_k	turbulent Prandtl numbers for k
σ_ε	turbulent Prandtl numbers for ε
Φ	general variable
Subscripts	
avg	mean value
eff	effective
t	turbulent
i,j	serial number

Habib, Ben-Mansour, Said, Al-Bagawi, and Al-Mansour (2008) and Habib, Ben-Mansour, Said, Al-Bagawi, and Al-Mansour (2009) conducted numerical investigations on different modified headers to optimize flow distribution inside the PFHE, the simulations indicated that flow distribution in the PFHE was greatly improved after applying the modified headers. Ismail, Ranganayakulu, and Shah (2009) numerically investigated the effects of nozzle and header orientation on the hydrodynamic performance of a PFHE, their studies showed that the orientation of the header and nozzle plays a major role in the exchanger performance.

However, due to the growing size and complicacy of the PFHE, it is not feasible to simulate actual full-size PFHE, which generally requires large computational resources and time. To our best knowledge, no study has yet been carried out to simulate a full-size PFHE by considering the actual number of fins inside the PFHE. Porous media model, which was proposed by Patankar and Spalding (1974), has been successfully employed in a variety of applications (Baker & Tabor, 2010; Barigou, Deshpande, & Wiggers, 2003; Lappalainen, Gorshkova, Manninen, & Alopaeus, 2011). By incorporating an empirically determined flow resistance in a region of the model defined as porous media, the flow hydrodynamic characteristics of the model can be considered equivalent to the actual geometry. The objective of this paper is to study the hydrodynamic characteristics inside a full-size PFHE by using CFD technique. Thus, a numerical PFHE model was proposed by adopting the porous media approach. The model was first validated through the experimental data. Based on the model, the hydrodynamic characteristics inside a full-size PFHE were then investigated. Finally, strategies for improving flow distribution of the PFHE were proposed and numerically verified.

2. Numerical simulation

2.1. Geometry of solution domain

Fig. 1 shows the geometry depiction of the solution domain, which is divided into three areas: the header area, the distribution area, and the heat transfer area. Fig. 2 shows the actual structure of a PFHE. It has a total of sixteen fin channels, with each fin channel corresponding to two distribution areas and one heat-transfer area. The fluid first flows through the inlet nozzle into the header, and then unevenly distributes into the sixteen fin channels according to the flow resistance of each fin channel. Both the distribution area and the heat transfer area are filled with perforated fins, as illustrated in Fig. 3. The main geometrical parameters of the perforated fins are listed in Table 1.

2.2. Simulation conditions and assumptions

The CFD code Fluent 6.3 was used to simulate the flow fields inside the PFHE. Mass flow inlet boundary condition was applied at the inlet of the model, and one atmospheric pressure was assigned to the outlet of the model. No-slip boundary and impermeability conditions were enforced at walls. In this study, inlet Reynolds number was in the range of 600–40,000 with reference length of 64 mm. Standard k–ε model with the enhanced wall function was adapted for turbulent modeling.

In this paper, only the hydrodynamic characteristics of the PFHE were considered. For simplicity, the following assumptions were applied: (1) only one side of the flow channel in the heat exchanger is modeled. (2) The fluid temperature is assumed as constant within the computational domain; (3) the working fluid is air, which behaves as ideal gas; (4) the flow is considered steady in the computational domain; (5) the wall is considered as ideal surface, which means there are no burrs, scarped edges, or adhesive substances on the wall.

In addition, since the fin channels of the PFHE are filled with perforated fins, it is prohibitively expensive and time-consuming to incorparte all fins owing to the limitation of the computational capability. Thus, we defined perforated fins in the channels as porous media, the effects of perforated fins on flow distribution and pressure drop of the PFHE can be obtained by setting viscous resistance and inertial resistance of that porous region. In this paper, a perforated fin model was first simulated to develop the porous formulation for the fin channel. Based on the porous formulation, the hydrodynamic characteristics of a full-size PFHE model was then studied by defining its fin channels as porous media.

2.3. Governing equations

Based on the above assumptions, a general governing equation used to describe the fluid flow in the model was established as follows:

$$\frac{\partial(\rho\Phi)}{\partial t} + \mathrm{div}(\rho U\Phi) = \mathrm{div}(\Gamma \mathrm{grad}\Phi) + S \qquad (1)$$