International Journal of Heat and Mass Transfer 55 (2012) 7561–7569

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical modeling and experimental validation of heat transfer and flow resistance on the shell side of a shell-and-tube heat exchanger with flower baffles

Yonghua You, Aiwu Fan *, Suyi Huang, Wei Liu

School of Energy and Power Engineering, Huazhong University of Science and Technology, Wuhan 430074, China

A R T I C L E I N F O

Article history:
Received 17 May 2012
Received in revised form 20 July 2012
Accepted 20 July 2012
Available online 19 August 2012

Keywords:
Shell-and-tube heat exchanger
Flower baffle
Numerical modeling
Experimental validation
Thermal augmentation
Flow resistance

A B S T R A C T

CFD simulation has become a powerful and popular tool for the thermal hydraulic design and analysis of heat exchangers. However, the computation load is usually too heavy to simulate a whole shell-and-tube heat exchanger (STHX) applying the traditional modeling method. In the present study, a numerical model based on the concepts of porosity and permeability is developed to obtain the shell-side thermal hydraulic performances. In this model, the distributed resistances and heat sources, as well as the distributed turbulence kinetic energy and its dissipation rate are introduced to account for the impacts of tubes on the fluid. The numerical model is solved over Re = 6813–22,326 for the shell side of a STHX with flower baffles, and reasonable accuracy is demonstrated by the comparison with test data (maximum relative deviation within 15%). With this model, the velocity and temperature fields, together with the distribution of convective heat transfer coefficient, are obtained and presented to help analyzing the underlying mechanism of shell-side thermal augmentation. The present work shows that this model is economic and effective in the thermal hydraulic design and analysis of a whole device.

1. Introduction

Shell-and-tube heat exchangers (STHXs) are widely applied in various industrial fields such as petroleum refining, power generation and chemical process, etc. Tremendous efforts have been made to improve the performances on the tube side [1–5]. For the shell side, the velocity and temperature fields are relatively complicated and the thermal hydraulic performance depends on the baffle elements to a great extent. In the traditional shell-and-tube heat exchangers with segmental baffles (SG-STHXs), round plates with cut are always used for the support of tubes [6–10]. Those baffles can also intensify the fluid flow which leads to heat transfer enhancement on the shell side. However, the flow resistance and harmful vibration level are large. Besides, the large-scale "dead" flow regions at the corners decrease the effective heat transfer area and facilitate the fouling on tubes. To solve these problems, various new types of baffles have been developed [11]. To give some examples, orifice baffles were designed to reduce the "dead" flow region and for heat transfer enhancement [12]. Rod baffles was originally developed to decrease the vibration level, which also showed good thermal hydraulic performances [13,14]. Helical baffles were proposed as the support of tubes and demonstrated good thermal hydraulic performance, low vibration level and less fouling as well [15,16]. In addition, ring support [17] and twisted self-supported tube [11] were also proposed and investigated.

Experimental test is a common method to investigate the thermal hydraulic performance on the shell side of heat-exchangers. However, experiments are relatively expensive and time-consuming. In addition, flow visualization on the shell side is also hard to deal with. Nowadays, the numerical method has become an economic alternative for the researches of STHXs, through which the detailed flow pattern and temperature field could be obtained with much less difficulties. Nevertheless, the computation load is always too heavy to model the whole device for desired information with reasonable accuracy. For this reason, a simplified model based on volumetric porosity and surface permeability was proposed and developed to model the shell-side flow and heat transfer [18–25]. With this method, the computation load decreases substantially while remaining a relatively high accuracy.

As mentioned above, STHXs with helical baffles have demonstrated a good overall performance on the shell side [26–28]. However, continuous helical baffles are difficult to manufacture, which hinders their wide applications. In order to induce spiral flow on the shell side while facilitate the manufacture, the STHX with flower baffles (FB-STHX) was designed in our group [29], in which round plates with two quadrants hollow were installed alternately in the shell as the support for tubes. And experimental investigations showed that this heat exchanger had a good thermal hydraulic

* Corresponding author. Address: School of Energy and Power Engineering, Huazhong University of Science and Technology, 1037 Luoyu Road, Wuhan 430074, China. Tel.: +86 27 87542618; fax: +86 27 87540724.

E-mail addresses: faw@mail.hust.edu.cn, catia315@163.com (A. Fan).

0017-9310/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.07.058

Nomenclature

A	heat transfer area, m^2
c_p	specific heat of constant pressure, J/(kg K)
d_i	inner diameter of tubes, m
d_o	outer diameter of tubes, m
D	inner diameter of shell, m
d_e	shell-side characteristic dimension, m
f	friction factor
f_S	surface permeability
f_V	volumetric porosity
h	convective heat transfer coefficient, W/(m^2 K)
k	coefficient, W/(m^2 K)
K	overall heat transfer coefficient, W/(m^2 K)
L	tube length, m
n	tube number
m	mass flow rate, kg/s
p	static pressure, Pa
Δp	static pressure loss, Pa
Pr	Prandtl number
Q	heat flux, W
Re	Reynolds number
t	time, s
T	static temperature, K
ΔT	temperature difference, K
u	velocity, m/s
x	coordinate axis, m
ρ	density, kg/m^3
μ	dynamic viscosity, N s/m^2
ε	turbulent kinetic energy dissipation rate, m^2/s^3
λ	thermal conductivity, W/(m K)
α	geometry angle of baffle, °
Ω	specific surface area, m^2/m^3

Subscripts

ex	exit
i, j	tensor
in	inlet
l	laminar
m	the mean value
s	shell-side
t	tube-side; turbulent

performance. In the present study, we are to develop a numerical model based on the concepts of porosity and permeability to obtain shell-side thermal hydraulic performances for the FB-STHX, and detailed flow and temperature fields will also be shown to help analyzing the underlying mechanisms. The numerical model will be validated with experimental data presented in [29,30].

2. Numerical model

A FB-STHX is of large dimension and complicated structure, thus the computation load is too large to obtain the flow and temperature fields on the shell side with traditional CFD modeling method, where the details of tube bundles need to be modeled and computed. For the sake of simplification, we borrow the concepts of volumetric porosity and surface permeability from porous media in the present work. The impacts of tubes on heat transfer and fluid flow on the shell side are considered by introducing a distributed heat source and a distributed flow resistance, which are determined by some correlations obtained through experiments. By doing this, the spaces occupied by tubes and the shell-side fluid could be meshed in the same grid system, which leads to substantial reduction in the cell number and computation load.

2.1. Governing equations on the shell side

In the present work, we are to compute the fluid flow and heat transfer on the shell side of the FB-STHX used in our previous experimental investigation [29], with the Re number ranging from 6813 to 22,326. The following assumptions were made for the derivation of governing equations:

(1) The working fluid on the shell side is continuous, incompressible, isotropic and Newtonian.
(2) Both the volumetric porosity and surface permeability are uniform in the tube-fluid mixed region; however, the impacts of tubes on fluid are non-uniform in different directions, which are reflected by the distributed resistances and heat source.
(3) The effect of gravity is negligible, and the viscous heating or thermal radiation is ignored.

Based on the above assumptions, the conservation equations of continuity, momentum and energy are presented below in a tensor form in the Cartesian coordinate system [19,20,23].

Continuity equation:

$$\frac{\partial(\rho f_V)}{\partial t}+\frac{\partial(\rho f_S u_j)}{\partial x_j}=0 \tag{1}$$

Momentum equation:

$$\frac{\partial(\rho f_V u_i)}{\partial t}+\frac{\partial(\rho f_S u_i u_j)}{\partial x_j}=-f_S\frac{\partial p_i}{\partial x_i}+\frac{\partial}{\partial x_j}\left(f_S\mu_{eff}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right) -f_i\frac{\rho}{2}|u_i|u_i \tag{2}$$

Energy equation:

$$\frac{\partial(\rho c_p f_V T)}{\partial t}+\frac{\partial(\rho c_p T f_S u_j)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(f_S\lambda_{eff}\left(\frac{\partial T}{\partial x_j}\right)\right)+\Omega K(T_t-T) \tag{3}$$

where all the parameters are for the shell-side fluid, except T_t that stands for the fluid temperature on the tube side. The variable u takes the formulation of physical velocity. The volumetric porosity, f_V, refers to the fraction of the void volume occupied by the fluid in a computation cell, while f_S stands for the surface permeability which equals to the fraction of the open projected flow area. Ω stands for the specific surface area of tube wall in the tube-fluid mixed region.

2.1.1. Distributed resistance on the shell side

The last term on the right hand side in Eq. (2), i.e., $-f_i\frac{\rho}{2}|u_i|u_i$, is the distributed flow resistance which accounts for the influence of tubes on the shell-side fluid flow, and is calculated through some empirical correlations. In this study, Eqs. (4) and (5) are adopted to calculate the cross-flow resistance factor in the staggered and non-staggered tube bundles, respectively [31], where the Re number is based on the cross-flow velocity component at the minimum flow area and the outer diameter of tubes acting as the characteristic dimension.

For staggered tube bundles:

$$f_i=4\left[0.23+\frac{0.11}{(\chi_j-1)^{1.08}}\right]\cdot Re_i^{-0.15}\cdot\left(\frac{f_S S_j}{S_j-d_o}\right)^2 \tag{4}$$