Numerical Simulation and Experimental Study of Flow and Heat Transfer Characteristics of Shell Side Fluid in Shell-and-Tube Heat Exchangers

Y.L.He¹, W.Q.Tao², B.Deng³, X.Li⁴ and Y.Wu⁵

1. Xi’an Jiaotong University, Xi’an, Shaanxi, China; yalinghe@mail.xjtu.edu.cn
2. Xi’an Jiaotong University, Xi’an, Shaanxi, China; wqtao@mail.xjtu.edu.cn
3. Xi’an Jiaotong University, Xi’an, Shaanxi, China; dengbin@mail.xjtu.edu.cn
4. Xi’an Jiaotong University, Xi’an, Shaanxi, China; Lizhiting_614@sohu.com
5. Xi’an Jiaotong University, Xi’an, Shaanxi, China; wuyangcom@sina.com

ABSTRACT

A three-dimensional, staggered grid, full-implicit control-volume based numerical model was presented for the analysis of laminar and turbulent fluid flow and heat transfer of shell side fluid in shell-and-tube heat exchangers. The numerical model used the distributed resistance method along with the concept of volumetric porosities, surface permeabilities to account for the presence of tubes in the heat exchangers. A modified $k-\varepsilon$ model was used to account for the effects of tubes on turbulence generation and dissipation. The wall function approach was used to deal with the near wall region. The three-dimensional model was validated by comparison of the computed pressure drop and heat transfer with some experiment data available obtained in the authors’ group. Good agreement between the simulation results and experimental data is obtained. It shows that applying the three-dimensional numerical model with the modified $k-\varepsilon$ model can effectively simulate the turbulence fluid flow and heat transfer in shell-and-tube heat exchangers. The anisotropic porosities model can simulate the flow characteristics in the shell-side of heat exchangers more effectively than the existing isotropic porous medium model.

INTRODUCTION

Shell-and-tube heat exchanger are widely used in process industries. Good understanding of the flow, and temperature fields within a heat exchangers is required for the design of reliable and efficient units. This information can be gained either by experimental testing (Bell, 1963; Pletcher and Andrews, 1994) or by simple analysis method (Tinker, 1958; Palen and Taborek, 1969). The methods of calculating the fluid resistance and heat transfer in the shell-and-tube heat exchangers, put forward by Tinker (1958) and Palen and Taborek (1969) are fully based on the correlations which drafted by the experiment data. However, the above methods can’t predict the detailed flow, temperature, and turbulence fields in heat exchangers, rather, they can only give overall pressure drop and heat transfer rate. The detailed field values of velocity, temperature, and turbulence intensity are crucial for a better understanding of heat exchanger operation, and for studies in the design of heat exchangers. Usually, experimental testing is very expensive and time consuming. In addition, flow visualization and detailed turbulence measurements in heat exchangers are difficult to perform. Therefore, numerical simulation provides a useful alternative and a properly validated numerical model for shell-and-tube heat exchangers can serve as a cost effective research tool.

The numerical research for heat exchangers of single-phase and two-phase fluids has been developing continuously, since the piniering work of Patankar and Spalding in 1974, and later enhanced by Butterworth (1978), Patankar (1984), Sha (1982, 1985), Prithiviraj and Andrews (1998a, 1998b), and so on. The numerical simulation of single-phase flow in the heat exchangers is the base for studying the even more complicated heat exchangers, such as the condenser, evaporator, etc.

The actual shell side fluid flow is so complicated that even for the present-day computational resource it is still prohibitive for most researchers to resolute the flow details by field numerical simulation. On the other hand, by using simplified model proposed by Patankar and Spalding (1974) we can still obtain some useful information while keep the requirement of computer resources in the limitation that is acceptable by engineering computation. The basic idea of Patankar and Spalding approach is viewing the shell side structure as a nominal porous medium and the presence of the tubes and baffles are taken into account by introducing parameters as volume porosity and surface permeability. In addition, the friction effect caused by the existence of the
solid region is taken into account by introducing empirical friction factor data as an additional source term in the momentum equation. This approach, even though relied on empirical friction data, provides an practical method to numerically simulate the shell-side performance and has been widely adopted since its publication.

In this paper, a three-dimensional, fully implicit, numerical method is used to simulate the shell-side single phase fluid flow and heat transfer. The simulation model is based on the distributed resistance concept along with a porous medium model. This model has been used to predict the shell-side performance of three kinds of shell-and-tube heat exchangers: bare tube banks with normal baffle, bare tube banks with helical baffle, and finned tube banks. All these research results have been compared with the experiment data available to the authors and reasonably good agreement between the simulation results and experimental data was obtained.

PHYSICAL MODEL

Porous Medium Model

Figure 1 shows a control volume that contains tubes. In the figure S is the total border surface area and \( S = S_f + S_s \), where \( S_f \) is the total fluid surface area. \( S_f = S_{f_s} + S_{f_g} \). \( S_{f_g}, S_{f_s}, S_{s} \) are the border between the fluid and fluid, the border between the fluid and solid, and that between solid and solid, respectively. Let \( V \) be the volume of the control volume, which includes fluid and solid parts, \( V_f \) the volume occupied by fluid in the control volume, and \( V_s \) the volume occupied by solid in the same control volume.

The volume porosity is a scalar quantity and can be determined as follows:

\[
fv = \frac{V_f}{V_f + V_s} = \frac{V_f}{V} \tag{1}
\]

The surface permeabilities is a vector to account for the fluid surface area blocked by the solid structure (Sha, 1982), whose three components in \( \vec{i}, \vec{j}, \vec{k} \) directions can be defined as follows:

\[
f_i = \frac{S_{f_g}}{(S_{f_g} + S_{s})} \vec{i}, f_j = \frac{S_{f_s}}{(S_{f_s} + S_{s})} \vec{j}, f_k = \frac{S_{f}}{(S_{f} + S_{s})} \vec{k} \tag{2}
\]

Governing Equations

For an arbitrary volume \( V \), with boundary surface \( S \), fluid density \( \rho \), surface flux \( J \) and body source \( S_\phi \) of any quantity \( \phi \), the general balance equation may be written as:

\[
\frac{d}{dt} \int_S \rho \phi dV = -\int_S \vec{n} \cdot \vec{J} dS + \int_{V_f} S_\phi dV \tag{3}
\]

By applying the Reynolds transport theorem and Gauss’ theorem to the Eq.(3), the following general differential balance equation can be obtained:

\[
\frac{\partial}{\partial t} (\rho \phi) + \nabla \cdot (f \rho \phi \vec{v} - f \Gamma_\phi \nabla \phi) = fS_\phi \tag{4}
\]

Integrating Eq. (4) over the fluid part of the control volume as shown in Fig. 1, and introducing the definition of volume porosity, surface permeabilities and distributed resistance, Eq. (4) can be simplified and expressed in the differential form as:

\[
\frac{\partial}{\partial t} (\rho \phi f) + \nabla \cdot (f \rho \phi \vec{v} - f \Gamma_\phi \nabla \phi) = fS_\phi \tag{5}
\]

where \( f \) in the first term of the left hand side and the right hand side is the volume porosity, while that in the divergence term is the surface permeability.

In the cylindrical coordinates, the corresponding governing equations for mass, momentum and energy can be expressed by Eq.(6), where the source terms in the momentum and energy equations are listed in Table 1. In the table, \( k \) and \( \varepsilon \) are the kinetic energy and dissipation rate of turbulent kinetic energy in the \( k-\varepsilon \) turbulence model. For the simplicity of presentation, the governing equations for \( k \) and \( \varepsilon \) are not shown in Eq.(6). The term \( T_s, T_i \) are the fluid temperatures of shell-side and tube side, and their source terms will be explained in the later discussion.

In the following the boundary conditions are illustrated for different kinds of the boundaries.
Table 1 Source terms in the governing equations

<table>
<thead>
<tr>
<th>φ</th>
<th>Γ_κ</th>
<th>S_κ</th>
</tr>
</thead>
<tbody>
<tr>
<td>u</td>
<td>η</td>
<td>( -\rho V^2 V + \frac{2}{r} \frac{\partial \rho V}{\partial \theta} )</td>
</tr>
<tr>
<td>v</td>
<td>η</td>
<td>( \rho V^2 )</td>
</tr>
<tr>
<td>w</td>
<td>η</td>
<td>( \frac{\partial \rho}{\partial r} + \rho g_z + R_z )</td>
</tr>
<tr>
<td>( T_s )</td>
<td>( \frac{\mu_k}{Pr} )</td>
<td>( \frac{\mu_k}{Pr} )</td>
</tr>
<tr>
<td>( T_l )</td>
<td>( \frac{\mu_\kappa}{\sigma_k} )</td>
<td>( G - \rho c + R_k )</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>( \frac{\mu_e}{\sigma_e} )</td>
<td>( C_1 \frac{V^2}{k} - C_2 \frac{c^2}{k} + R_z )</td>
</tr>
</tbody>
</table>

Inlet Boundary Condition

Take a shell and tube heat exchanger with normal baffles as an example (Fig. 2). The oncoming velocity \( U_{in} \) is known to be vertically downward, and the inlet domain is the section where the shell surface and the inlet tube surface intersect, which is characterized by \( \theta < \theta_0 \), and \( r < r_{in} \). For this region the three velocity components can be determined as follows:

\[
\begin{align*}
    u &= u_{in} \cdot \sin \theta \\
    v &= -u_{in} \cdot \cos \theta \\
    w &= 0
\end{align*}
\]

The inlet temperatures of shell side fluid and tube side fluid are pre-specified. The inlet kinetic energy for turbulent flow is taken as 1% of the kinetic energy of the oncoming flow and the dissipation \( \varepsilon \) is determined by following two equations:

\[
2 \rho u_{in} R_{in} / \eta_i = 500
\]

\[
\eta_i = C_u \rho k^2 / \varepsilon
\]

Outlet Boundary Condition

The definition of the outlet region is the same as the inlet region except for the axial location. For the velocity component which is normal upward, the mass conservation condition is used to determine the outlet value from the adjacent inner point. The details for this treatment can be found in Tao (2001). At the outlet the other two components are treated as fully developed, i.e., their first derivatives with respect to radius equals zero. The same treatment is adopted for the shell side fluid temperature, kinetic energy and the dissipation term.

Symmetry Surface

On the symmetric surface where \( \theta = \pi \) following condition applies:

\[
\begin{align*}
    u &= 0, \quad \frac{\partial V}{\partial \theta} = 0 \\
    \frac{\partial w}{\partial \theta} &= 0
\end{align*}
\]

Solid Wall

On all solid walls for velocity no slip boundary condition is applied, and for temperature the adiabatic condition is adopted.

Distributed Resistances

Based on the flow in anisotropic porosity media, under the cylindrical coordinates, distributed resistance can be defined as three parts: radius flow \( R_r \), circumferential flow \( R_\theta \) and axial flow \( R_z \). For the bare tube banks, Zukauskas conducted experiments on tubes in cross flow and obtained pressure correlations. Base on these test data, the distributed resistances \( R_r \) and \( R_\theta \) are formulated as:

\[
R_n = \frac{\Delta p}{\Delta r} = 0.5 \frac{1}{\Delta r} N_i \rho FC^2 V_{r_{max}}^3
\]
\[
R_n = \left( \frac{\Delta p}{r \Delta \theta} \right)_{p} = 0.5 \frac{1}{r \Delta \theta} N_t \rho CV^2 \theta_{\text{max}}
\]  

(11)

where \(\Delta r, \Delta \theta\) are the dimensions of the computational cell. \(N_t\) is the number of tubes in cross flow in a given direction \(r\) or \(\theta\). \(\rho\) is the density of the fluid, \(V_{\text{max}}\) is the fluid velocity of the minimum area of cross section in the \(r\) direction and \(\theta\) direction respectively. \(f\) is the friction factor, and \(x\) is a geometry factor dependent on the arrangement of tubes. Zukahauskas (1988) provided the charts for \(f\) and \(x\). As for the friction along tube axis, the pressure drop correlations of Rehme (1973) can be adopted:

\[
R_z = 0.5 \frac{1}{A} f \rho V_z^2
\]

(12)

where the friction factor is determined by

\[
\left( \frac{8}{f} \right)^{0.5} = A \left[ 2.5 \ln(\text{Re} \left( \frac{f}{8} \right)^{0.5}) + 5.5 \right] - G^* \]

(13)

where \(A\) and \(G^*\) are geometry parameters given by Rehme(1973) which depend on the tube layout. \(\text{Re}\) is the Reynolds number based on the hydraulic diameter of the computational cell.

Tubes in the heat exchangers enhance the production and dissipation of turbulence kinetic energy. According to, Prithiviraj and Andrews(1998a,1998b) the turbulence generation due to the existence of tubes can be formulated as:

\[
R_i = \int R \cdot V dV = R_s V_s + R_\theta V_\theta + R_r V_r
\]

(14)

The resistance source of turbulence dissipation is deduced from a limiting case of a fully developed flow in a tube bundle as( Prithiviraj and Andrews, 1998a,1998b):

\[
R_s = 1.92 \frac{e}{k} R_s
\]

(15)

where \(R_s, R_\theta\) and \(R_r\) are given by Equations (10),(11) and (12).

The Treatment of Tubes Bundles and Baffle in the Simulation of Temperature Field

In the simulation of shell-and-tube heat exchangers, the shell envelope can be considered as insulator. The energy conservation can be described that the heat lost by the shell-side fluid is gained by the tube-side fluid or, alternately, the heat lost by the tube-side fluid is gained by the shell-side fluid. In the distributed resistance approach, the shell-side fluid in a computational cell has an enthalpy \(H_s\) and the tube-side fluid has an enthalpy \(H_t\). The heat source \(\phi\) for a given computational cell is computed by integrating the heat source or sinks within the computational cell over the surface area of tubes \(A_s\). The heat transfer can be given as:

\[
\phi = \int_{A_s} k \left( \frac{H_t}{C_{p,t}} - \frac{H_s}{C_{p,s}} \right) dA
\]

(16)

where \(k\) is the overall heat transfer coefficient, \(C_{p,s}, C_{p,t}\) are the shell-side and tube-side fluid specific capacity, respectively. The source term for the energy equation is obtained by integrating Equation (16):

\[
S_f = kA_s \left( \frac{H_t}{C_{p,t}} - \frac{H_s}{C_{p,s}} \right) = kA_s (T_t - T_s)
\]

(17)

As for the tube-side, it is assumed that the tube-side flow in a shell-and-tube heat exchanger is fully developed. Since the flow within the tubes is in the axial direction and diffusion is negligible, the governing equation for tube-side flow is:

\[
\frac{\partial}{\partial z} \left( \rho \frac{dH}{dz} \right) = \rho C_{p,t} \frac{dT}{dz}
\]

(18)

The overall heat transfer coefficient, \(k\), can be obtained from shell-side heat transfer coefficient and the tube side coefficient which can be found from Zukauskas (1988) for bare tube banks and from Gnielinski (1977). Then for bare tube banks the overall heat transfer coefficient can be calculated by:

\[
k = \frac{1}{\left( \frac{1}{R_s} + \left( \frac{d_0}{d_0 - 2 \delta_{s}} \right) \left( \frac{1}{R_\theta} \right) + \left( \frac{d_0}{d_0 - 2 \delta_{s}} \right)^2 \right)}
\]

(19)

where the overall heat transfer coefficient is based on the outer surface area of the tube banks.

**NUMERICAL SIMULATION METHODS**

The governing equations are discretized by the finite volume method. The convective term is discretized by the power-law scheme. The coupling between velocity and pressure is deal with by the SIMPLER algorithm based on a stagger grid, where scalar properties such as pressure and enthalpy are calculated at the center of computational cells, while cell faces are used for each of the velocity components. The resulting algebraic equations are solved by the alternative direction iteration method. The velocity fields are first solved and then is the temperature field.

Numerical simulations were conducted for following three heat exchangers:

1. HEx with vertical baffles;
2. HEx with helical baffles;
3. HEx with finned tube banks.

In the following, the numerical results will be presented for the three heat exchangers in order, and comparisons with experimental data will be made whenever possible.
NUMERICAL RESULTS OF HEX WITH VERTICAL BAFFLES

Simulation Conditions and Definition of Shell-side Re

The simulation conditions are shown in Table 2.

Table 2 Simulation conditions for HEx with vertical baffles

<table>
<thead>
<tr>
<th>No</th>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Shell-side fluid</td>
<td>air</td>
</tr>
<tr>
<td>2</td>
<td>Tube-side fluid</td>
<td>water</td>
</tr>
<tr>
<td>3</td>
<td>Shell diameter</td>
<td>0.41m</td>
</tr>
<tr>
<td>4</td>
<td>Shell length</td>
<td>1.98m</td>
</tr>
<tr>
<td>5</td>
<td>Number of baffle</td>
<td>3, 5, 7, 9</td>
</tr>
<tr>
<td>6</td>
<td>Baffle cut</td>
<td>25%</td>
</tr>
<tr>
<td>7</td>
<td>Tube diameter</td>
<td>0.015m</td>
</tr>
<tr>
<td>8</td>
<td>Number of tubes</td>
<td>325</td>
</tr>
<tr>
<td>9</td>
<td>Inlet/outlet diameter</td>
<td>0.148m</td>
</tr>
<tr>
<td>10</td>
<td>Tube side inlet temperature</td>
<td>353K</td>
</tr>
<tr>
<td>11</td>
<td>Shell-side inlet temperature</td>
<td>293K</td>
</tr>
</tbody>
</table>

A shell-side Reynolds number is defined as follows:

\[
\text{Re}_s = \frac{G_C \cdot d_t}{\eta} = \frac{\frac{1}{4} \pi D_a^2 \cdot \rho \cdot u_{in} \cdot d_t}{[d_t - (\frac{d_t}{P_t})d_t] \cdot \frac{L}{N_{cross}}} \cdot \frac{d_t}{\eta}
\]

where \(G_C\) (kg/m²s) is the mass velocity of the shell-side fluid in the minimum area of cross section along the centerline of the heat exchanger, \(P_t\) is the tube pitch and \(N_{cross} = N + 1\).

Velocity, Temperature and Pressure Fields

Simulations were conducted for tube side Reynolds number of 2000 and shell side Reynolds number varying from 10000 to 100000. For \(Re_s = 50000\), the predicted velocity, pressure and temperature fields in the longitudinal cross section for the five baffles case are presented in Figs. 3 and 4, and the three fields in different axial cross sections are shown in Fig. 5, where \(z = 0.22\) m and 1.75 m are the cross sections of inlet and exit respectively. As seen in the figs., the fluid enters the heat exchanger at the inlet and sweeps back and forth across the tubes before it leaves out through the exit. The shell-side fluid velocities are high in the inlet and outlet regions because of the sudden expansion and contraction. For the case studied, no recirculation zones are formed near the baffles. For the case studied the pressure variation range is 150000Pa and the fluid temperature varies from 294K to 340K.

The Effect of Re on Pressure Drop and Heat Transfer

In the two cross sections of \(z = 0.22\) m and 0.8 m, the main flow goes from shell top to the bottom. Thus the shell side fluid temperature reaches its maximum at the bottom of the cross section as seen in Fig. 5(b).
For the case of five baffles, the variation of the shell side total pressure drop with $Re_s$ is shown in Fig. 6. The results can be well fitting by following equations:

$$\Delta p = 4.78 \times 10^{-4} Re_s^{1.814}$$

(17)

This is in good agreement with fluid mechanics in that for the turbulent flow fluid pressure drop is proportional to the $u^{1.75}$ to $u^{2.0}$.

**Effect of Baffle Number on $\Delta p_{shell}$ and $\Delta T_{shell}$**

Figure 8 shows that the effect of baffle number on pressure drop distribution in the heat exchanger under the same shell-side mass flow rate ($U_{in} = 102.7$ m/s). As seen in the figure, under the same shell-side mass flow rate, with the increase in baffle number, the pressure drop between the inlet and outlet increases. However, in the regions of sudden expand and contract, the local resistance is the same for different baffle case because of the same mass flow rate and geometry structure. When the baffle number is small, the major pressure drop occurs in the inlet and outlet region.

Figure 9 shows the variation of shell side section average temperature difference between outlet and inlet with the baffle number. It can be seen that for a fixed shell side mass flow rate, the total temperature difference increases with the baffle number, but the increase tendency gradually decreases. For the case studied, nine baffles are probably the maximum, beyond which increase in baffle
number will lead to the increase in pressure drop while no appreciable gain in heat transfer can be obtained.

Experimental Validation of Pressure Drop Data

Three models of TEMA-E heat exchanger were made by plexiglass. The main characteristics of this heat exchanger is its simplicity: single shell and single tube side. The experimental apparatus is shown in Fig 10. In the heat exchanger model eleven pressure measuring taps were set up along the axial length, including the inlet and outlet sections. Air was used as the working fluid. The measured pressure drop variations along the axial length for six mass flow rate are presented in Fig. 11 for the case of 7 baffles. It can be seen that the agreement between predicted and measured pressure drop is quite satisfactory. The relative deviations are listed in Table 3. The maximum deviation occurs at the largest mass flow rate. It can be observed that the predicted pressure drops all larger than that of measured. This is because in the practical model there are flow leaks through different paths, while in our numerical models these leaks are not taken into account.
(d) Mass flow rate = 0.31 m³/s

(e) Mass flow rate = 0.33 m³/s

(f) Mass flow rate = 0.34 m³/s

Fig. 10 Comparisons of predicted and measured pressure drops for heat exchanger with 7 baffles

Fig. 11 Effect of baffle number on the total pressured drop

The effect of baffle number on total pressure drop is shown in Fig. 11 for predicted and measured data at the mass flow rate of 0.24 m³/s. The maximum deviation is 20.5%.

From above comparisons it can be concluded that the numerical methods and the code developed can predict flow characteristics with a reasonable good agreement.

In the following presentations, numerical results for shell and tube heat exchangers with helical baffles and finned tube banks will be presented. The major numerical methods and the same as that for the vertical baffles, hence will not be re-stated. Only those special numerical treatments for the individual case will be briefly mentioned.

NUMERICAL RESULTS OF HEX WITH HELICAL BAFFLES

Special Numerical Treatments

Because the helical configuration, the symmetry condition at the central longitudinal section can not be adopted anymore, rather, the whole region of the cylindrical space should be taken as the computational domain. In addition to meet the requirement for simulation of different baffle configurations, the most simple but useful method to deal with an irregular domain - stepwise approximation is adopted to simulate the baffle.

Reynolds Number Definition

In order to compare flow and heat transfer performance of heat exchanger with vertical and helical baffles, the shell-side Reynolds number is defined as follows:

\[
Re = \frac{ud_s}{v}
\]  

(18)
where the characteristic dimension $d_e$ takes the value of the tube diameter, $u$ is the flow velocity in the minimum section, which is defined by Eq.(19):

$$u = \frac{q_{mo}}{\rho_o A_m}$$  \hspace{1cm} (19)

In Eq.(19), $A_m$ is the area of the minimum flow section, and $q_{mo}$ is the shell side mass flow rate. The minimum flow area is determined by following equations for vertical baffles and helical baffles:

- **Vertical baffle**: $A_{m,ver} = Bd_e(1 - \frac{d_o}{p_t})$  \hspace{1cm} (20a)
- **Helical baffle**: $A_{m,hel} = \frac{1}{2} Bd_e(1 - \frac{d_o}{p_t})$  \hspace{1cm} (20b)

where $B$ is the distance between two adjacent baffles, $d_e$ is the diameter of the shell, $d_o$ is the diameter of the heat transfer tube, and $p_t$ is the tube spacing.

It is to be noted that the above definition of Re makes the numerical results comparable between those of vertical baffles and of helical baffles.

**Helical Configurations Simulated**

There are a lot of configurations of the helical baffle (B. Deng, 2003). The most fundamental configurations are presented in Figs. 12 and 13. For each figure the top picture is a pictorial view of the helical baffles and the bottom one is its simple presentation. In is study, the configurations shown by Fig. 12 and Fig.13(b) were simulated.

**Simulation Results of 1st Combination of Fluids**

Simulation conditions are presented in Tables 4 and 5. For the simplicity of presentation, only the results of staggered half-ellipse baffle will be presented.

**Table 4** Simulations conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shell-side fluid</td>
<td>Air</td>
</tr>
<tr>
<td>Tube-side fluid</td>
<td>Water</td>
</tr>
<tr>
<td>Shell length</td>
<td>2.0 m</td>
</tr>
<tr>
<td>Shell diameter</td>
<td>0.6 m</td>
</tr>
<tr>
<td>Tube diameter</td>
<td>0.015 m</td>
</tr>
<tr>
<td>Tube pitch</td>
<td>0.020 m</td>
</tr>
<tr>
<td>Number of tubes</td>
<td>325</td>
</tr>
<tr>
<td>Diameter of inlet/outlet</td>
<td>0.20 m</td>
</tr>
</tbody>
</table>

**Table 5** Baffle parameter

<table>
<thead>
<tr>
<th>Baffle</th>
<th>Baffle pitch, B, m</th>
<th>Baffle number</th>
<th>Helical angle, degree</th>
</tr>
</thead>
<tbody>
<tr>
<td>Helical ,,Fig. 12(a)</td>
<td>0.272</td>
<td>20</td>
<td>12,15,18</td>
</tr>
<tr>
<td>Helical ,,Fig. 12(b)</td>
<td>0.180</td>
<td>28</td>
<td>12,15,18</td>
</tr>
<tr>
<td>Vertical</td>
<td>0.140</td>
<td>9</td>
<td>0</td>
</tr>
</tbody>
</table>
The velocity distribution in a longitudinal cross section which transversely cuts the two half-ellipse baffles is presented in Fig. 14. As it can be seen in the figure, the fluid flow route is helical. The fluid flows through the empty space between the baffles, rushes to the shell wall of the heat exchangers, and part of it turns over to the other half of the heat exchangers then flows to the next circle.

In order to reveal the effect of helical angle on the total pressure drop, simulations were performed for three helical angles shown in Table 5, and the results are provided in Figs. 15, where the results for the vertical baffle are also presented for comparison. It can be clearly observed that compared with the shell-and-tube heat exchanger with vertical baffle the helical baffle can greatly reduce the pressure drop. And the larger the helical angle the smaller the pressure drop.

Simulation and Test Results of 2nd Combination of Fluids

Two series of experimental measurements were conducted for the staggered sector helical baffle(Fig.13(b)), one for measurement of shell side pressure drop without heat transfer and the other for the measurement of heat transfer. The conditions of the two series of tests are summarized in Tables 6 and 7. The test system for heat transfer measurement is presented in Fig.16. For the measurement of pressure drop, only the oil loop was used. To take advantage of the test data, numerical simulations were performed accordingly.

Table 6 Test condition for pressure drop measurement

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length of heat exchangers</td>
<td>720mm</td>
</tr>
<tr>
<td>Diameter of shell</td>
<td>140mm</td>
</tr>
<tr>
<td>Diameter of tubes</td>
<td>10mm</td>
</tr>
<tr>
<td>Tube pitch</td>
<td>15mm</td>
</tr>
<tr>
<td>Number of tubes</td>
<td>69</td>
</tr>
<tr>
<td>Configuration of tube</td>
<td>square</td>
</tr>
<tr>
<td>arrangement</td>
<td></td>
</tr>
<tr>
<td>Diameter of inlet tube</td>
<td>40mm</td>
</tr>
<tr>
<td>Diameter of outlet tube</td>
<td>40mm</td>
</tr>
<tr>
<td>Baffle pitch</td>
<td>68.4mm</td>
</tr>
<tr>
<td>Number of baffles</td>
<td>40</td>
</tr>
</tbody>
</table>

Table 7 Test conditions for heat transfer measurement

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shell diameter</td>
<td>182 mm</td>
</tr>
<tr>
<td>O/I tube diameter</td>
<td>10 / 7mm</td>
</tr>
<tr>
<td>Tube arrangement</td>
<td>Square</td>
</tr>
<tr>
<td>Tube thermal conductivity</td>
<td>36.7 W/m·K</td>
</tr>
<tr>
<td>Length of heat exchanger</td>
<td>1550 mm</td>
</tr>
<tr>
<td>Active length of tube</td>
<td>1139 mm</td>
</tr>
<tr>
<td>Tube number</td>
<td>72</td>
</tr>
<tr>
<td>Oil inlet tube ID</td>
<td>51 mm</td>
</tr>
<tr>
<td>Oil outlet tube ID</td>
<td>51 mm</td>
</tr>
</tbody>
</table>

Fig. 16 Test system for heat transfer measurement

The comparison of pressure drops from measurement and simulation is shown in Fig. 17. It can be seen that within the mass flow rate of oil studied (2.0 m³/h to 4.0 m³/h) the agreement of the predicted and test data is quite good, with the maximum relative deviation of 18%.
The measured and predicted data for heat transfer are presented in Table 7, where for five data points, the predicted values of the outlet temperatures of the oil and water are listed. The number in the parentheses are the relative deviation based on temperature difference. It can be found that the agreement is quite good.

**NUMERICAL RESULTS OF HEX WITH FINNED TUBE BANKS**

Usually bare tubes are used in shell-and-tube heat exchangers. However, when the shell side fluid is air enhancement technique must be adopted to reduce the heat transfer surfaces. The most often adopted technique is to use finned tube bank, with an integrated fin sheet shared with a certain number of tubes. Such configuration is often used when the heat exchanger is used to cool air or gas with high pressure. Figure 18 presents such a heat exchanger which is widely used in Chinese factories as an air/gas intercooler. Numerical simulation was also performed with some special treatment of the shell side distributed resistance.

In Fig. 18(b) the shaded area is the blocked region so that the air steam go through the center part where finned tube banks are located. As far as the water side is concerned, two-pass arrangement was made in order to improve the local overall temperature difference distribution over the entire heat exchanger. This is shown in Fig. 19.
implemented by setting $R_z = \infty$ (a very large number) in Eq.(12). In addition, no any baffles exist in the shell side. In the discretization of the computational domain, one has to be care of the axial control volume arrangement. Since the fin spacing is usually in the order of several millimeters, and the length of the heat exchanger is in meters, it is often impossible from computer memory consideration that each control volume only contains one piece of fin sheet. Often five to ten pieces of fin sheet should be included in one control volume in order to make the resulting algebraic equations manageable by a personal computer.

**Determination of Distributed Resistances**

From Eqs.(10), (11) it can be seen that in order to calculate the distributed resistances, $R_\theta, R_\tau$, the value of friction factor should be known. For the finned tube banks such as the one shown in Fig. 19, the general correlation resulted from bare tube banks can not be used. To this end, special test was arranged to obtain the friction factor correlation. Following result was obtained:

$$\lg f = 7.3389 - 3.6004 \lg(Re_{\text{max}}) + 0.42125(\lg Re_{\text{max}})^2 \quad (21)$$

where $Re_{\text{max}}$ is defined as follows:

$$Re_{\text{max}} = \frac{u_{\text{max}} d}{\nu} \quad (22)$$

In Eq.(22) $u_{\text{max}}$ is the maximum average gas velocity occurring at the minimum flow area.

**Simulation Conditions**

Simulation conditions were copied from a practical air intercooler which are listed in Table 8. As indicated in the table there are 870 pieces of fin sheet. *7 control volumes were used in the finned tube region such that each control volume includes 10 pieces of fin sheets.

<table>
<thead>
<tr>
<th>Table 8 Simulation conditions for finned tube HEx</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Parameter</strong></td>
</tr>
<tr>
<td>Shell-side fluid</td>
</tr>
<tr>
<td>Tube-side fluid</td>
</tr>
<tr>
<td>Shell diameter</td>
</tr>
<tr>
<td>Shell length</td>
</tr>
<tr>
<td>Fin thickness</td>
</tr>
<tr>
<td>Fin spacing</td>
</tr>
<tr>
<td>Fin Number</td>
</tr>
<tr>
<td>Tube diameter</td>
</tr>
<tr>
<td>Tube spacing</td>
</tr>
<tr>
<td>Tube number</td>
</tr>
</tbody>
</table>

**Velocity, Pressure and Temperature Distributions**

For $Re_{\text{max}} = 20000$ and tube side $Re = 10000$, the predicted velocity, pressure and temperature distributions in the longitudinal symmetric section are shown in Figs. 20 and 21. In the figures, the relative variation range of pressure is 0-80000 Pa, and the variation range of air temperature is 347 K ~ 353 K. It can be observed following features. (1) In the middle cross section the air pressure drop is the largest because the highest downward air velocity in this region. (2) The cooling effect of the middle part of the finned tube

![Fig. 20 Velocity and pressure distributions](image1)

![Fig. 21 Velocity and temperature distributions](image2)

is the worst. Although in the middle part the air side heat transfer coefficient is the largest compared with other part of the heat exchanger, the enhancement extent of heat transfer can not compensate the largest local flow rate of the air. Thus even though the local heat transfer rate is larger than anywhere else in the heat exchanger, the local exit temperature of the air is the largest. This result and that mentioned above for pressure drop may be an indication that the air distribution in the inlet region of the heat exchanger needs to be improved such that in the exit region the uniformity of air temperature and pressure can be better.

The effect of shell side Reynolds number on the total pressure drop is presented in Fig. 22 for the case of tube side Reynolds number = 10000. By carefully inspecting the figure, it can be found that the increase of pressure drop with the increase in shell side $Re$ is very significant, much faster than of the flow in ducts and across tube banks. As indicated in the figure the increase in flow rate from $Re=10000$ to $Re=20000$ leads to an increase in pressure drop from around $17000$ Pa to $80000$ Pa. Estimated by these two data, the exponent in the Reynolds number is about 3.5 to 3.6, instead of 1.75 to 2.0 (see Eq.(17)). This may be accounted by the fact that the existence of the fin sheet greatly increases the frictional resistance to the flow across the finned tube bank.
The effect of shell side Reynolds number on the heat transfer rate is shown in Fig. 23. It can be seen that the heat transfer rate increases with Re almost linearly, implying that even for the finned tube banks the airside thermal resistance is still a dominated component in the total thermal resistance compared with the water side resistance component.

CONCLUSIONS

In this keynote lecture for numerically predict heat transfer and flow characteristics of shell side fluid in shell-and-tube heat exchangers, a modified porous medium model in conjunction with distributed resistance and two-equation $k−\varepsilon$ closure was presented in detail. The model and developed codes were used to simulate three kinds of shell-and-tube heat exchangers (vertical baffles, helical baffles and finned tube banks), and comparisons were made between the predicted results and test data specially obtained in the authors group to verify the model. Generally speaking reasonably good agreements were received. Although such model requires that the friction factor correlation is known for the shell side fluid, it can still provide useful information for the shell side fluid flow and heat transfer and is a useful tool in the design of heat exchangers.

Further research needs to be performed includes the accumulation of more reliable friction factor correlations. In the numerical algorithm aspect, development of a new computational system which possesses a much faster convergence rate with enough numerical accuracy and stability is highly required.

ACKNOWLEDGMENTS

The work reported here was supported by following foundations: National Natural Foundation of China (No. 50476046, 50425620, 50236010); A Foundation for the Author of National Excellent Doctorial Dissertation of P R China; and RFDP20030698015.

REFERENCES

Bell K J. Final report of the cooperative research program on shell-and-tube heat exchangers. Bulletin No. 5. University of delaware engineering experimental station, Newark, Delaware, 1963


Deng B. Experimental and numerical study of flow and heat transfer in the shell side of heat exchangers. Ph D. Thesis. Xi’an Jiaotong University, Xi’an, China, 2003


Tao W.Q. Numerical heat transfer. 2nd edition. Xi’an Jiaotong University Press, Xi’an, 2001


Zukauskas A. Convective heat transfer in heat exchangers. Science Press, Moscow, 1982