A comparison of four numerical modeling approaches for enhanced shell-and-tube heat exchangers with experimental validation

Jie Yang a,b, Lei Ma a, Jessica Bock b, Anthony M. Jacobi b, Wei Liu a,*

a School of Energy and Power Engineering, Huazhong University of Science and Technology, Wuhan 430074, China
b Mechanical Science and Engineering, University of Illinois, Urbana, IL, USA

HIGHLIGHTS

• Four different modeling approaches are developed for rod baffles heat exchanger.
• Heat transfer and pressure drop of computational calculations are studied.
• Experimental validation is carried out to verify the numerical results.
• The comparisons of four various modeling approaches are conducted.

ARTICLE INFO

Article history:
Received 18 September 2013
Accepted 19 January 2014
Available online 28 January 2014

Keywords:
Rod-baffle heat exchanger
Turbulence
Numerical simulation
Experimental validation

ABSTRACT

In the present paper, 3-D numerical simulations of a rod-baffle shell-and-tube heat exchanger with four different modeling approaches are developed and validated with experimental results. The four methods of modeling include two in which a small subsection of the heat exchanger is modeled (the unit model, and the periodic model), one in which the heat exchanger is consider as a porous medium (the porous model), and one in which the entire heat exchanger is modeled with CFD (the whole model). The results illustrate that the periodic model, porous model and whole model can have high accuracy in predicting heat transfer, while the unit model has relatively low accuracy. The porous model and whole model also provide good predictions of the pressure drop, but the unit model and periodic model fail to accurately predict pressure drop. The porous model requires accurate heat transfer correlations for the heat exchanger, and such correlations may not be available for new designs. The whole model demands significant computational resources for geometric modeling, grid generation, and numerical calculation. A demonstration of different grid systems for various models is also conducted. In summary, the present work provides a comparison of various modeling approaches and an analysis of trade-offs between numerical accuracy and computational demands for models of shell-and-tube heat exchangers.

© 2014 Elsevier Ltd. All rights reserved.

1. Introduction

Shell-and-tube heat exchangers are widely used in the petrochemical industry, manufacturing industry, food preservation, electrical power production and energy conservation systems, due to their structural simplicity, relatively low cost and design adaptability. According to Master and co-workers, they account for more than 35%~40% of the heat exchangers used in global heat transfer processes [1]. Many novel structures, such as segmental baffles [2], double segmental baffles [3], multi-segmental baffles [4], orifice baffles [5], rod baffle [6], self-braced structure with twisted tubes [7], helical baffles [8] and ring baffles [9] have been suggested to enhance heat transfer and reduce pumping power. Among the above-mentioned enhancement techniques, the rod-baffle heat exchanger exhibits excellent thermal-hydraulic performance and was selected for the current study. The rod-baffle design was first proposed by Phillips Petroleum Company in 1976 to solve flow-induced vibration of tube bundles, fouling and corrosion problems of shell-side [10]; the design has found broad application with advantages of higher heat transfer efficiency, lower pressure loss, larger effective heat transfer area, better anti-fouling performance and so on compared to the segmented baffle heat exchanger [11–13].

Experiments can provide highly reliable measurements of thermal-hydraulic performance; however, experiments can be
dicting heat transfer coefficient and pressure drop, even though the model did not take the inlet and outlet nozzles and frictional resistance of inner shell wall into account. Liu et al. [16] applied the unit model for a new longitudinal flow shell-and-tube heat exchanger (the rod-vane compound baffle heat exchanger) and compared to the rod-baffle heat exchanger. The results demonstrated that the unit model is applicable for longitudinal flow heat exchanger. You et al. [17] used the unit model of a trefoil-hole baffled shell-and-tube heat exchanger to visually analyze the flow field and one working point was quantitatively compared to experimental data, which displayed a good accuracy in thermal-hydraulic performance. The periodic model [18–20] regards the flow as periodically fully developed by setting the inlet and outlet as periodic boundary conditions. The periodic model also neglects the inlet and outlet effects of the shell-side. Gu et al. [18] studied the symmetrically periodic model for segmented-baffle and rod-baffle heat exchanger to overcome some insufficiencies of the previous modeling methods, but the results were not validated by any correlations or experiments. Lei et al. [19] conducted simulations of a helical-baffle heat exchanger with different baffle inclination angles using a periodic model as the helically baffle heat exchangers are periodic along the flow direction. A comparison of numerical and experimental results was carried out by examining the average Nusselt number and friction factor, and the results indicated a good consistency. Zhang et al. [20] carried out performance predictions of a heat exchanger with middle-overlapped helical baffles with different helix angles using the simplified periodic model, and they analyzed the pressure drop inconsistency between simulation and experiment, mainly resulting from inlet and outlet nozzles. The model with volume porosity, surface permeability and distributed flow resistances [21–24] regards flow zones as porous media without heat transfer tubes, so that a relatively coarse grid can be used. The porous model eliminates tube bundles and achieves accuracy by introducing volume porosity and surface permeability as parameters, adding distributed heat sources and distributed flow resistances that are determined from empirical correlations. Obviously, the porous model is not suitable.

There are four main modeling approaches used for numerical simulation: the unit model, the periodic model, the porous model and the whole model. The unit model [15–17] regards the flow zone in shell-and-tube heat exchangers as a unit flow channel. As shown in Fig. 1, the unit flow channel is restricted by four virtual boundary walls and four quarter-tubes walls. In most cases, the virtual walls are set as symmetric boundary conditions and the tube walls are set as solid-wall boundary conditions. Obviously, the unit model ignores the inlet and outlet nozzles effects. Dong et al. [15] simulated a flow unit duct model of rod-baffle heat exchanger and conducted comparisons to both correlations and experiments. The results indicated that the model demonstrated a high precision in predicting heat transfer coefficient and pressure drop, even though the extremely expensive and time-consuming compared to computational fluid dynamics (CFD). Unfortunately, for very complex flows, such as those prevailing in the rod-baffle shell-and-tube heat exchanger, selecting an appropriate modeling approach can be difficult. There are complex tradeoffs between accuracy and computational expense. A heat exchanger with 500 heat transfer tubes and 10 baffles requires at least 150 million computational cells to resolve the geometry [14].

Research [15–29] about different models has been conducted. There are four main modeling approaches used for numerical simulation: the unit model, the periodic model, the porous model and the whole model. The unit model [15–17] regards the flow zone in shell-and-tube heat exchangers as a unit flow channel. As shown in Fig. 1, the unit flow channel is restricted by four virtual boundary walls and four quarter-tube walls. In most cases, the virtual walls are set as symmetric boundary conditions and the tube walls are set as solid-wall boundary conditions. Obviously, the unit model ignores the inlet and outlet nozzles effects. Dong et al. [15] simulated a flow unit duct model of rod-baffle heat exchanger and conducted comparisons to both correlations and experiments. The results indicated that the model demonstrated a high precision in predicting heat transfer coefficient and pressure drop, even though the model did not take the inlet and outlet nozzles and frictional resistance of inner shell wall into account. Liu et al. [16] applied the unit model for a new longitudinal flow shell-and-tube heat exchanger (the rod-vane compound baffle heat exchanger) and compared to the rod-baffle heat exchanger. The results demonstrated that the unit model is applicable for longitudinal flow heat exchanger. You et al. [17] used the unit model of a trefoil-hole baffled shell-and-tube heat exchanger to visually analyze the flow field and one working point was quantitatively compared to experimental data, which displayed a good accuracy in thermal-hydraulic performance. The periodic model [18–20] regards the flow as periodically fully developed by setting the inlet and outlet as periodic boundary conditions. The periodic model also neglects the inlet and outlet effects of the shell-side. Gu et al. [18] studied the symmetrically periodic model for segmented-baffle and rod-baffle heat exchanger to overcome some insufficiencies of the previous modeling methods, but the results were not validated by any correlations or experiments. Lei et al. [19] conducted simulations of a helical-baffle heat exchanger with different baffle inclination angles using a periodic model as the helically baffle heat exchangers are periodic along the flow direction. A comparison of numerical and experimental results was carried out by examining the average Nusselt number and friction factor, and the results indicated a good consistency. Zhang et al. [20] carried out performance predictions of a heat exchanger with middle-overlapped helical baffles with different helix angles using the simplified periodic model, and they analyzed the pressure drop inconsistency between simulation and experiment, mainly resulting from inlet and outlet nozzles. The model with volume porosity, surface permeability and distributed flow resistances [21–24] regards flow zones as porous media without heat transfer tubes, so that a relatively coarse grid can be used. The porous model eliminates tube bundles and achieves accuracy by introducing volume porosity and surface permeability as parameters, adding distributed heat sources and distributed flow resistances that are determined from empirical correlations. Obviously, the porous model is not suitable.

---

**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>hydraulic area (m²)</td>
</tr>
<tr>
<td>$c_p$</td>
<td>specific heat capacity (kJ kg⁻¹ K⁻¹)</td>
</tr>
<tr>
<td>$C$</td>
<td>empirical constant (–)</td>
</tr>
<tr>
<td>$d_i$</td>
<td>inner tube diameter (m)</td>
</tr>
<tr>
<td>$d_o$</td>
<td>outer tube diameter (m)</td>
</tr>
<tr>
<td>$D$</td>
<td>hydraulic diameter (m)</td>
</tr>
<tr>
<td>$D_s$</td>
<td>inner shell diameter (m)</td>
</tr>
<tr>
<td>$F$</td>
<td>LMTD correction factor (–)</td>
</tr>
<tr>
<td>$f_v$</td>
<td>overall volume porosity (–)</td>
</tr>
<tr>
<td>$f_{vo}$</td>
<td>local volume porosity (–)</td>
</tr>
<tr>
<td>$G$</td>
<td>producing term (kg m⁻¹ s⁻¹)</td>
</tr>
<tr>
<td>$H$</td>
<td>heat transfer coefficient (W m⁻² K⁻¹)</td>
</tr>
<tr>
<td>$K$</td>
<td>turbulent kinetic energy (m² s⁻²)</td>
</tr>
<tr>
<td>$m$</td>
<td>mass flow rate (kg s⁻¹)</td>
</tr>
<tr>
<td>$n$</td>
<td>tube quantity (–)</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt number (–)</td>
</tr>
<tr>
<td>$Q$</td>
<td>heat transfer capacity (J)</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number (–)</td>
</tr>
<tr>
<td>$S_k$</td>
<td>additional term of $k$ (kg m⁻¹ s⁻¹)</td>
</tr>
<tr>
<td>$S_\varepsilon$</td>
<td>additional term of $\varepsilon$ (m² s⁻¹)</td>
</tr>
<tr>
<td>$t$</td>
<td>time (s)</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature (K)</td>
</tr>
</tbody>
</table>

**Subscripts**

- $\text{ave}$: average value
- $c$: cold water
- $ci$: inlet cold water
- $co$: outlet cold water
- $\varepsilon$: dissipation rate term
- $\text{eff}$: effective term
- $f$: fluid
- $h$: hot water
- $hi$: inlet hot water
- $ho$: outlet hot water
- $i, j, k$: tensor
- $x, y, z$: coordinate axis

---

![Fig. 1. The unit model.](image-url)
for heat exchangers where there are no correlations available. Prithviraj et al. [14,21] numerically investigated a conventional shell-and-tube heat exchanger using the distributed resistance approach, which regards that every single computational cell may have multiple tube bundles leading to a relatively coarse grid system. Sha et al. [22] improved this porous model by adding surface permeability and volumetric porosity. By doing this, the porous model can obtain higher precision while keep the computational resources in an acceptable limit. Later He et al. [23] modified the porous model in conjunction with \(k-\varepsilon\) two-equation and extended the application of the porous model into three kinds of heat exchangers (vertical baffles, helical baffles, and finned tube banks). Recently Shi et al. [24] performed a semi-porous media approach using a large-scale sparse tubular heat exchanger and compared it to the conventional porous model with the experimental validation. The results showed that the semi-porous model can predict air to the conventional porous model with the experimental validation. The applications of different helically baffled heat exchangers with various geometries [25–28] demonstrated that high precision of thermal-hydraulic predictions of the whole model was achieved by a large grid system. Ozden et al. [28] numerically investigated the effects of different turbulent models on a small segmental baffle shell--and-tube heat exchanger with nine tubes due to the limitations of computational resources with the emphasis on choosing the proper mesh density, order of discretization, and turbulence model. The novelty and significance of the above research rest in providing four different numerical approaches, illustrating implementation process of the modeling methods, and demonstrating wide case applications. Once a new heat transfer enhancement technique towards heat exchangers application is invented, certain criteria are required to evaluate the performance and effectiveness. So far, the performance evaluation criteria (PEC) [29–39] have been proposed and successfully utilized to analyze and evaluate the thermal-hydraulic performance.

Although the realization of four different models is well established, a criterion of how to select the most appropriate numerical approach according to corresponding circumstances has not been built yet. In other words, despite much numerical research on shell--and-tube heat exchangers, a trade-off between numerical accuracy and computational resources has seldom been considered in the above literature review. As a matter of fact, a comprehensive comparison about thermal-hydraulic performance accuracy, consumed computational resources, and applicability of the above-mentioned four different modeling approaches has not been found in the open literature; such a comparison is especially important for large-scale heat exchangers [40]. Therefore it is necessary to compare different models and elucidate the advantages and disadvantages of the approaches currently available to heat exchanger designers. In this paper, four different models, the unit model, the periodic model, the porous model, and the whole model of a rod-baffle shell-and-tube heat exchanger, are adopted for computational study and an experimental system is used to validate and assess the numerical results. The present work conducts a comparison on numerical accuracy, grid system size, computational period, and restriction for four different models, analyzing trade-offs between computational accuracy and demands, providing the advantages and disadvantages of each modeling approach to academic researchers and engineering designers, and filling the existing gap in the open literature.

## 2. Physical model

### 2.1. Model introduction

The geometrical configuration of the rod-baffle heat exchanger for four different models is presented in Fig. 2. Rod baffles are manufactured from an array of support rods that are welded at each end to a circumferential baffle ring, and a variable number of baffles are placed along the flow direction in alternating orientation (vertically arranged, horizontally arranged, vertically arranged again, etc) so that, together, they fix tubs in fixed locations. As seen in Fig. 2, the rod baffles are perpendicular to each other to brace the heat transfer tubes. For the unit model, the unit flow channel is formed by the outer walls of four tubes and a square boundary wall. Several cylinders arranged in a perpendicular way are inserted in the flow zone as rod baffles. For the periodic model, the flow channel is formed with one complete rod baffle placed between tube bundles. For the porous model, the exchanger with rod baffles, the inlet and outlet of the shell-side is modeled without the presence of tubes bundles. For the whole model, a detailed heat exchanger with heat transfer tubes, the inlet and outlet for the shell-side, connecting rods and rod baffles are modeled. All parameters of numerical simulations are identical with the experimental setup. Regular \(k-\varepsilon\) equations are adopted for turbulent flow zone, and these equations will be expressed in the following content.

### 2.2. Governing equations, grid generation and boundary conditions

#### 2.2.1. Governing equations for regular models

The conservation equations for the regular computational models are presented in the tensor form in the Cartesian coordinate system as the flow is steady and the fluid is incompressible [41,42].

**Continuity equation:**

\[
\frac{\partial u_i}{\partial x_j} = 0 \quad (1)
\]

**Momentum equation:**

\[
\rho \left( \frac{\partial (u_i u_j)}{\partial x_j} \right) = - \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right) \quad (2)
\]

**Energy equation:**

\[
\rho \left( \frac{\partial (u_i T)}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left( \frac{\lambda}{c_p} \frac{\partial T}{\partial x_j} \right) \quad (3)
\]

**\(k\)-equation:**

\[
\rho \left( \frac{\partial (k u_i)}{\partial x_j} \right) = \frac{\partial}{\partial x_j} \left( \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + G_k - \rho \varepsilon \quad (4)
\]

**\(\varepsilon\)-equation:**

\[
\frac{\partial (\varepsilon u_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\varepsilon}{C_1} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{C_{1,\varepsilon}}{k} G_k - C_2 \rho \varepsilon^2 \quad (5)
\]

where \(\rho\) is fluid density, \(\lambda\) is thermal conductivity, \(c_p\) is specific heat capacity, \(p\) is pressure, \(\mu\) is the dynamic viscosity, \(k\) is turbulent kinetic energy, \(\varepsilon\) is turbulent dissipation rate, \(G_k\) is producing term of turbulent kinetic energy generated by mean velocity gradient, \(C_1\) and \(C_2\) are empirical constants, \(\sigma_k\) and \(\sigma_\varepsilon\) are Prandtl numbers corresponding to turbulent kinetic energy and turbulent dissipation rate, \(\mu_t\) is defined as follows:
\[
\mu_t = \rho C_m k^2
\]  
(6)

where \( C_m = 0.09, C_1 = 1.44, C_2 = 1.92, \sigma = 1.0, \sigma_t = 1.3 \) and \( G_k \) is defined as follows:

\[
G_k = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}
\]  
(7)

### 2.2.2. Governing equations for the model based on volume porosity, surface permeability and distributed flow resistances

The conservation equations for the computational model on the basis of volume porosity, surface permeability and distributed flow resistances are presented in the tensor form in the Cartesian coordinate system [21–24,42].

Continuity equation:

\[
\frac{\partial u_j}{\partial x_j} = 0
\]  
(8)

Momentum equation:

\[
\rho \frac{\partial (f_i u_i u_j)}{\partial x_j} = \frac{\partial (f_i p_i)}{\partial x_j} + \frac{\partial}{\partial x_j} \left( f_i \mu_{\text{eff}} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right) - f_i \frac{p}{2} |u_i| u_i
\]  
(9)

Energy equation:

\[
\rho \frac{\partial (f_i u_i) T}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \lambda_{\text{eff}} \frac{\partial T}{\partial x_j} \right) + \frac{\Omega q}{C_p}
\]  
(10)

\( k \)-equation:

\[
\rho \frac{\partial (f_i k u_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( f_i \left( \mu + \frac{\mu_t}{\sigma} \right) \frac{\partial k}{\partial x_j} \right) + f_i (G_k - \rho \varepsilon) + S_k
\]  
(11)

\( \varepsilon \)-equation:

\[
\rho \frac{\partial (f_i \varepsilon u_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( f_i \left( \mu + \frac{\mu_t}{\sigma} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + f_i \left( \frac{C_1 \varepsilon}{K} G_k - C_2 \rho \varepsilon^2 \right) + S_{\varepsilon}
\]  
(12)

where \( f_i \) stands for the volume porosity which equals to ratio of fluid volume to fluid-plus-solid volume on a control volume, \( f_s \) stands for the surface permeability which is equal to the ratio of fluid area to fluid-plus-solid area on a control volume face, \( \Omega \) stands for the specific surface area of tube wall in the fluid-solid mixed region, \( q \) is the internal heat source due to the absence of heat transfer tubes, \( \mu_{\text{eff}} \) and \( \lambda_{\text{eff}} \) are the effective dynamic viscosity and the effective thermal conductivity, which are equal to the sum of laminar and turbulent counterparts. \( S_k \) and \( S_{\varepsilon} \) are the additional...
the absence of tube bundles, respectively, and defined as follows:

\[ S_k = \frac{1}{2} \sum_i f_i \rho |u_i|^3 \]  

(13)

\[ S_r = \frac{1.92}{2k} \sum_i f_i \rho |u_i|^3 \]  

(14)

The last term in Equation (9) \( f_i \rho |u_i|/2 \) is the distributed flow resistance which accounts for the influence of heat transfer tubes. The friction coefficient \( f_i \) in the term is calculated by empirical correlations as follows:

\[ f_x = 4 \left( \frac{C_x}{\eta} \right) \left( \frac{\eta f_x}{\eta - D} \right)^2 \left( 1 - f_{x0} \right) \left( 1 - f_x \right) \]  

(15)

\[ f_y = 4 \left( \frac{C_y}{\eta} \right) \left( \frac{\eta f_y}{\eta - D} \right)^2 \left( 1 - f_{y0} \right) \left( 1 - f_y \right) \]  

(16)

\[ f_z = 4 \left( \frac{C_z}{\eta} \right) \left( \frac{1 - f_{z0}}{1 - f_z} \right) \]  

(17)

where \( D \) is the hydraulic diameter for shell-side, \( \eta \) is tube pitch, \( f_x \) and \( f_{x0} \) are overall and local volume porosity factors, respectively. The local volume porosity factor \( f_{x0} \) is defined as:

\[ f_{x0} = 1 - \frac{\pi}{4} \left( \frac{d_0}{\eta} \right)^2 \]  

(18)

where \( d_0 \) is the outer diameter for tube and friction factors \( C_x \), \( C_y \) and \( C_z \) are given as follows:

For cross flow:

\[ C_x = \begin{cases} 
0.619 - \frac{Re_x^{0.198}}{Re_x^{0.2647}} & \text{if } Re_x < 8000 \\
1.156 - \frac{Re_x^{0.198}}{Re_x^{0.4297}} & \text{if } 8000 \leq Re_x < 2 \times 10^5 
\end{cases} \]  

(19)

\[ C_y = \begin{cases} 
0.619 - \frac{Re_y^{0.198}}{Re_y^{0.2647}} & \text{if } Re_x < 8000 \\
1.156 - \frac{Re_y^{0.198}}{Re_y^{0.4297}} & \text{if } 8000 \leq Re_x < 2 \times 10^5 
\end{cases} \]  

(20)

For parallel flow:

\[ C_z = \begin{cases} 
31/Re_z & \text{if } Re_x < 2250 \\
0.131 - \frac{Re_z^{0.249}}{Re_z^{0.4267}} & \text{if } 2250 \leq Re_x < 25000 \\
0.066 - \frac{Re_z^{0.227}}{Re_z^{0.4267}} & \text{if } Re_x \geq 25000 
\end{cases} \]  

(21)

where Reynolds number is based on the velocity components and the outer diameter of tube.

2.2.3. Grid generation

The geometrical modeling and grid generation procedures are carried out with commercial CFD preprocessor GAMBIT 2.3. The grid independence test was completed for each model. Taking the unit model as an example, five different grid systems with \( 5.0 \times 10^4, 1.9 \times 10^5, 3.8 \times 10^5, 5.3 \times 10^5 \) and \( 7.5 \times 10^5 \) cells are adopted for calculation. The differences in heat transfer coefficient and pressure drop between the third and fourth model are around 7% and the differences between the fourth and fifth model are around 2%. Thus, taking numerical resource cost and solution accuracy into consideration, the fourth model with \( 5.3 \times 10^5 \) cells was adopted. The grid diagrams for the unit model, periodic model, porous model and whole model are presented in Figs. 3-6. Due to the complex structure of rod baffle, the models are discretized with hexahedral meshes for the most flow region and with tetrahedral
Fig. 5. Meshes of porous model.

Fig. 6. Meshes of whole model.
meshes for the rod baffle region. After the grid independence test, the final cell numbers for the unit model, periodic model, porous model and whole model are $5.3 \times 10^5$, $1.4 \times 10^6$, $3.6 \times 10^6$ and $2.5 \times 10^7$, respectively.

The commercial CDF software Fluent 6.3 was adopted for all the numerical simulations. The 3-D, double-precision, pressure-based solver was used. The conservation equations are discretized with a finite volume formulation. In particular, for the model on the basis of volume porosity, surface permeability and distributed flow resistances, the influences of absence of tubes on the conservation equations of momentum, energy, turbulence kinetic energy and turbulence kinetic energy dissipation rate are treated by introducing the corresponding source terms with six user defined functions (UDFs). The corresponding UDFs are programmed by Microsoft Visual C++, and the program is in the Appendix for convenience.

2.2.4. Boundary conditions

The standard wall function method is adopted for the near-wall region, and the non-slip boundary condition is adopted on all solid surfaces. The surfaces of the rod baffles are set as adiabatic because the impact caused by thermal conduction of the rod baffle can be neglected, and taking the rod baffle surface as adiabatic allows for a coarser grid. The inner wall of shell-side is also set as adiabatic because the heat exchanger is thermally isolated during the experiment. The finite difference method and the second-order upwind difference scheme are applied, and the SIMPLE algorithm is adopted for the coupling between pressure and velocity fields.

The second-order upwind difference scheme is applied for energy and momentum computation, and the standard difference scheme is used for the pressure. For the unit model, porous model, and whole model, the velocity-inlet boundary condition is applied for the inlet since for incompressible fluid the velocity-inlet boundary condition is equal to mass-flow-inlet boundary condition in Fluent, and the outflow boundary condition is applied for the outlet since the pressure for outlet is not given. The periodic boundary condition is applied for inlet and outlet for the periodic model. The temperatures of the tube wall and inlet fluid are set as constants for the unit model, periodic model and whole model. The four boundary walls of the unit model are set as symmetry boundary conditions. The other setting parameters adopted default settings according to user’s guide in Fluent. Water is set as the working fluid, and the corresponding parameters are presented in Table 1. The following assumptions are made to simplify numerical simulations: the thermal—physical properties of the fluid such as $\rho$, $\mu$, $c_p$, $\lambda$ are constant; the working fluid is incompressible, isotropic, Newtonian and continuous; the effect of gravity is negligible and viscous heating and thermal radiation are ignored.

3. Experimental apparatus

3.1. Experimental table and test section

The schematic for the experimental system is presented in Fig. 7. The system consists of three loops, which are hot water loop, cooling water loop and refrigerating loop. The hot water loop contains a 58 kW electrical heater, water tank, hot water pump, flow meters and tube-side of rod-baffle heat exchanger. The cooling water loop contains the water side of a plate heat exchanger, water tank, cold water pump, flow meters and the shell-side of a rod-baffle heat exchanger. The refrigerating loop contains a 58 kW refrigerating unit device, pump and the refrigerant side of a plate heat exchanger. When the experiment is in steady operation, the heat generated by electrical heater is transferred from the hot water to the cold water in the rod-baffle heat exchanger. Then the thermal

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_p$ (J/kg K)</td>
<td>4182</td>
</tr>
<tr>
<td>$\mu$ (kg/m s)</td>
<td>0.001003</td>
</tr>
<tr>
<td>$\rho$ (kg/m³)</td>
<td>998.2</td>
</tr>
<tr>
<td>$\lambda$ (W/m K)</td>
<td>0.6</td>
</tr>
</tbody>
</table>

Fig. 7. Schematic diagram for experimental system.
power in cooling water loop transfers heat from water to refrigerant at the plate heat exchanger. Finally, heat is rejected to ambient air. The volume flow rates for cold and hot water are measured using four rotary flow meters. The temperature of the water is measured using four K-type thermal couples that are inserted into holes at the hot water inlet, hot water outlet, cold water inlet, and cold water outlet, as shown in Fig. 7. The pressure drops between inlet and outlet for the shell and the tube-side are measured using the two pressure drop transmitters. All data of temperature and pressure difference are transmitted in the PC system and automatically recorded through a data acquisition system. Experiments are conducted with various volume flow rates ranging from 5.5 to 19.1 m$^3$/h since the measuring range for the rotary flow meters is 2.0–20 m$^3$/h. The temperature ranges from 18.2 to 27.7 °C for shell-side and 37.4–44.4 °C for tube-side.

The test section of the rod-baffle heat exchanger is shown in Fig. 8. The dimensional parameters for the exchanger and rod baffle are presented in Fig. 9 and Table 2. All thermal physical parameters for water such as density, thermal conductivity and viscosity are adopted at the average temperature of inlet and outlet and parameters for stainless steel are adopted at the average temperature of cold and hot water. The heat transfer from tube-side is approximately equal to the heat transfer to the shell-side since the shell is thermally insulated. During the experiment, all parameters are measured when the experiment is in steady operation for 10 min; namely, the energy balance deviation between hot and cold water is less than 5% for each measurement that is defined as follows:

$$\eta = \frac{|Q_h - Q_c|}{Q_{\text{min}}} \times 100\% \leq 5\% \quad (22)$$

The experimental parameters in steady performance are listed in Table 3.

### 3.2. Data reduction

The Reynolds number for the shell-side is expressed as follows:

$$Re_s = \frac{\rho_s u_s d_s}{\mu_s} \quad (23)$$

where $u_s$ is the velocity for shell-side in the cross-section where there are no baffles, and is expressed as follows:

$$u_s = \frac{m_s}{\rho \left( \frac{D^2}{4} - \pi \cdot n \cdot d_o^2 \right)} \quad (24)$$

and $D$ is expressed as follows:

$$D = \frac{\pi \cdot D_s^2 - \pi \cdot n \cdot d_o^2}{\pi \cdot D_s + \pi \cdot n \cdot d_o} \quad (25)$$

where $D_s$ is inner shell-side diameter and $n$ is tube quantity. The Reynolds number for the tube-side is expressed as follows:

$$Re_t = \frac{\rho_t u_t d_i}{\mu_t} \quad (26)$$

where $u_t$ is the velocity for tube-side and $d_i$ is the inner diameter of tube. Nusselt number for shell-side is calculated as follows:

$$Nu_s = \frac{h_s \cdot D}{\lambda_f} \quad (27)$$

where $h_s$ is the convective heat transfer coefficient for shell-side and $\lambda_f$ is thermal conductivity for fluid.
For experimental research, the convective heat transfer coefficient for shell-side $h_t$ is calculated as follows:

$$h_t = \frac{N_u \cdot \lambda_f}{d_i}$$

(28)

where $\lambda_f$ is thermal conductivity for the fluid, and tube-side Nusselt number $N_u$ is calculated using Gnielinski equation [45]. The overall heat transfer coefficient $K$ is calculated as follows:

$$K = \frac{Q}{A \Delta t_m \cdot F}$$

(30)

where $A$ is heat transfer area based on the outer diameter of heat transfer tubes, $Q$ is heat transfer amount and $\Delta t_m$ is the log-mean temperature difference, $F$ is the log-mean temperature difference correction factor according to TEMA standards [46].

As the uncertainties of $Q_h$ and $Q_c$ are quantitatively close, the heat transfer amount $Q$ is calculated as follows for simplicity:

$$Q = \frac{Q_h + Q_c}{2} = \frac{m_h \cdot C_h(T_{hi} - T_{ho}) + m_c \cdot C_c(T_{co} - T_{ci})}{2}$$

(31)

where $Q_h$ and $Q_c$ are heat transfer amount for hot and cold water, $m_h$ and $m_c$ are mass flow rate for hot and cold water, $C_h$ and $C_c$ are heat capacity for hot and cold water, $T_{hi}$, $T_{ho}$, $T_{co}$ and $T_{ci}$ are the heat inlet temperature, hot water outlet temperature, cold water inlet temperature and cold water outlet temperature, respectively.

It should be noticed that the arithmetic mean heat transfer amount is not preferred if the experimental uncertainties for both streams are not identical, as Jacobi et al. presented in Ref. [47].

The log-mean temperature difference (LMTD) between hot and cold water is calculated as follows:

$$\Delta t_m = \frac{(T_{hi} - T_{co}) - (T_{ho} - T_{ci})}{\ln \frac{T_{hi} - T_{ho}}{T_{co} - T_{ci}}}$$

(32)

For numerical simulations, the convective heat transfer coefficient for shell-side $h_t$ is calculated as follows:

$$h_t = \frac{Q}{A \cdot \Delta t_m \cdot F}$$

(33)

where $Q$ is heat transfer amount, $A$ is heat transfer area, $T_w$ is temperature for tube wall and $T_{ave}$ is average temperature for the working fluid. The above four parameters may vary in different computational cases. As illustrated in Refs. [48], the difference between arithmetic mean temperature difference (AMTD) between $T_w$ and $T_{ave}$ is close (4%) to LMTD when $(T_w - T_{in})/(T_w - T_{out}) \leq 2$. In fact, the results in this work using different temperature differences are almost identical (less than 1%). Therefore, the latter expression in Equation (33) is adopted to calculate the heat transfer coefficients for simplicity.

3.3. Uncertainty in experimental data

The test range and accuracy of measurements are given in Table 4. The uncertainties of the experimental data are calculated with the method of reference [49]. It involves calculating variables derivatives with respect to each experimental quantity and applies already known uncertainties, which is calculated as follows:

$$U_R = \sqrt{\sum_{i=1}^{n} \left( \frac{\partial R}{\partial x_i} U_{x_i} \right)^2}$$

(34)

where $R = f(x_1, x_2, \ldots, x_n)$ represents a desired variable, $x_1, x_2, \ldots, x_n$ are the variables that impact $R$, and their absolute uncertainties are expressed by $U_{x_1}, U_{x_2}, \ldots, U_{x_n}$, respectively. After calculation, the uncertainties for the Reynolds number, the total heat transfer, the overall heat transfer coefficient, the shell-side heat transfer coefficient, the Nusselt number, and the pressure drop for both the shell- and tube-side are 0.26%–0.74%, 0.62%–1.16%, 4.64%–5.69%, 4.64%–5.69%, 4.64%–5.69% and 0.99%–5.89%, respectively, when the Reynolds number ranges from 14,000 to 5000 as the large Reynolds number corresponds to small uncertainty in Table 5.

4. Analysis of numerical and experimental results

In this paper, the computational simulations were conducted under the same conditions as the experiments. The comparisons of Nusselt number for four numerical models and experiment are presented in Fig. 10. It is clearly seen that the Nusselt number trends of all data are similar, that is, Nusselt number increases with Reynolds number. For the unit model, it is noted that the heat transfer coefficient is relatively smaller than the experimental data. This is mainly because the boundary is set as symmetry wall condition, thus there is no heat and mass transfer on that boundary in

<table>
<thead>
<tr>
<th>Table 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structural parameters.</td>
</tr>
<tr>
<td>Material</td>
</tr>
<tr>
<td>Shell outer diameter</td>
</tr>
<tr>
<td>Shell inner diameter</td>
</tr>
<tr>
<td>Tube outer diameter</td>
</tr>
<tr>
<td>Tube inner diameter</td>
</tr>
<tr>
<td>Tube effective length</td>
</tr>
<tr>
<td>Tube pitch</td>
</tr>
<tr>
<td>Tube number</td>
</tr>
<tr>
<td>Baffle number</td>
</tr>
<tr>
<td>Baffle thickness</td>
</tr>
<tr>
<td>Rod diameter for baffle</td>
</tr>
<tr>
<td>Baffle pitch</td>
</tr>
<tr>
<td>Baffle distance from head</td>
</tr>
<tr>
<td>Tube arrangement</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow for hot water</td>
</tr>
<tr>
<td>Flow for cold water</td>
</tr>
<tr>
<td>Inlet temperature for hot water</td>
</tr>
<tr>
<td>Outlet temperature for hot water</td>
</tr>
<tr>
<td>Inlet temperature for cold water</td>
</tr>
<tr>
<td>Outlet temperature for cold water</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test range and accuracy of instruments.</td>
</tr>
<tr>
<td>Instruments</td>
</tr>
<tr>
<td>K-type thermocouples</td>
</tr>
<tr>
<td>Rotary flow meters for shell-side</td>
</tr>
<tr>
<td>Rotary flow meters for tube-side</td>
</tr>
<tr>
<td>Pressure drop transmitter for shell-side</td>
</tr>
</tbody>
</table>
numerical calculation, while in fact there is plenty of turbulence and secondary flow leading to heat and mass transfer on the virtual boundary in the experiment. The unit model also ignores the impingent effects on the tube bundle caused by the inlet flow on the shell-side. Therefore, the unit model is preferable to be used for visual analysis rather than quantitative analysis to some degree [11]. For the periodic model, it is noted that the Nusselt number is smaller than experimental values for some data. But when the Reynolds number exceeds 11,000, the Nusselt number for the periodic model is larger than that of the experiments. It is expected that the periodic model is more suitable for heat exchangers of relatively large length as the inlet and outlet impingent effects would be neglected in that case. For the porous model, it is noted that the variation tendencies for numerical and experimental Nusselt numbers are very consistent. Quantitatively, the deviation of Nusselt number between calculation and experiment is 6.5–12.4%, which shows that the porous model has a relatively high accuracy for heat transfer predictions. For the whole model, the values of Nusselt number are almost identical to experiments for relatively low Reynolds number. The deviation from experimental data increases with the Reynolds number. However, the Nusselt number trends for the whole model are still in good agreement with experiments.

A comparison of pressure drop for the unit model and periodic model is presented in Fig. 11. The pressure drop in Fig. 10 is calculated by multiplying the pressure drop of each unit or periodical region by the number of repetitions. For the unit model there are twelve complete unit flow regions for the whole heat exchanger, and for the periodical model there are six complete periodical flow regions for the whole heat exchanger. It should be noticed that the frictional pressure drop for inner shell wall, inlet and outlet nozzles of shell-side is not taken into consideration for the unit model, and the pressure drop for the inlet and outlet nozzles is not taken consideration for the periodical model. It is seen that the pressure drop for the unit flow channel is approximately 20% smaller than the experimental data. It is also seen that the pressure drop for the periodical model is approximately 40% larger than the experimental data. In Ref. [20], a method is provided to calculate the total pressure drop, that is, multiply the cycle number by pressure drop in one cycle and add the pressure drop caused by the inlet and outlet nozzles according to Gaddis and Gnielinski correlations [50].

As presented in Fig. 12, the trends in pressure drop for the porous model and whole model are similar. The pressure drop for the porous model is smaller than experimental data, while that for the whole model is larger. Quantitatively, the deviation of pressure drop between calculation and experiment is 3.6–12.1% for the porous model and the difference is 0.7–12.4% for the whole model.

Numerical simulations for four methods of modeling differ in geometry, flow fields, results and computational costs. The comparison for different models with various cell numbers is given in Table 6. It is concluded that the porous model and whole model have high accuracy on predicting heat transfer, pressure drop and overall performance but the porous model requires extra codes and known correlations for source terms; the whole model requires large computational resources. It is also concluded that the unit model has low accuracy on predicting thermal characteristics and is unable to predict pressure drop due to much simplifications. It is also concluded that the periodical model has high accuracy on heat

![Fig. 10. Comparison of shell-side Nusselt number for different models and experiment.](image1)

![Fig. 11. Comparison of shell-side pressure drop for unit model and periodic model.](image2)

![Fig. 12. Comparison of shell-side pressure drop for porous model, whole model and experiment.](image3)

<table>
<thead>
<tr>
<th>Objects</th>
<th>Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reynolds number</td>
<td>±(0.26%–0.74%)</td>
</tr>
<tr>
<td>Heat transfer amount</td>
<td>±(0.62%–1.16%)</td>
</tr>
<tr>
<td>Overall heat transfer coefficient</td>
<td>±(4.64%–5.69%)</td>
</tr>
<tr>
<td>Shell-side heat transfer coefficient</td>
<td>±(4.64%–5.69%)</td>
</tr>
<tr>
<td>Nusselt number</td>
<td>±(4.64%–5.69%)</td>
</tr>
<tr>
<td>Pressure drop</td>
<td>±(0.99%–5.89%)</td>
</tr>
</tbody>
</table>
transfer prediction and is more suitable for exchanger of larger length. However it is unable to directly predict pressure drop.

5. Conclusions

In the present paper, four different modeling approaches of a rod-baffle heat exchanger are developed, which are the unit model, the periodic model, the porous model, and the whole model. The CFD software GAMBIT is used for the geometrical modeling and grid generation procedures and FLUENT is used for all computational calculations. An experimental system is built to validate the results. The present work provides a comparison of the four numerical methods to fill the gap in open literature, the main conclusions of which are as follows:

1. The experiments validate the precision of each model in predicting heat transfer and pressure drop with the experimental validation. It is concluded that the periodic model, porous model and whole model have a high accuracy on predicting heat transfer performance, while the unit model has a relatively low accuracy; it is concluded that the porous model and whole model has high accuracy on predicting pressure drop, while unit model and periodic model are unable to directly predict hydraulic performance.

2. Trade-offs between computational resources and accuracy are analyzed. The whole model with highest accuracy consumes largest resources on geometrical modeling, grid generation, and numerical calculations; the porous model with relatively high precision consumes medium numerical resources, however, it requires extra codes and is not applicable for new designs; the periodic model consumes relatively low resources and the unit model consumes the lowest.

Acknowledgements

This work is supported by the National Basic Research Program of China (Grant No. 2013CB228302) and the National Natural Science Foundation of China (Grant No. 51036003) and the China Scholarship Council.

Table 6

<table>
<thead>
<tr>
<th></th>
<th>Unit model</th>
<th>Periodic model</th>
<th>Porous model</th>
<th>Whole model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cell number</td>
<td>530,000</td>
<td>1,400,000</td>
<td>3,600,000</td>
<td>25,000,000</td>
</tr>
<tr>
<td>Precision of heat transfer</td>
<td>Low</td>
<td>High</td>
<td>High</td>
<td>High</td>
</tr>
<tr>
<td>Precision of pressure drop</td>
<td>x</td>
<td>High</td>
<td>High</td>
<td>High</td>
</tr>
<tr>
<td>Time cost</td>
<td>Short</td>
<td>Short</td>
<td>Medium</td>
<td>Long</td>
</tr>
<tr>
<td>Restriction</td>
<td>None</td>
<td>None</td>
<td>Codes and correlations</td>
<td>None</td>
</tr>
</tbody>
</table>

Appendix

#include "udf.h"

double Cx,Cy,Cz; /* empirical correlations */

double Density_01,Viscosity_01,Diameter_01; /* thermal physical property */

double Coefficient_01,Coefficient_02; /* corresponding coefficient */

DEFINE_SOURCE(x_momentum_source,cell,thread,dS,eqn)
{
    real source;
    if(Density_01*fabs(C_U(cell,thread))*Diameter_01/Viscosity_01<8000)
        Cx=0.619*pow(Density_01*fabs(C_U(cell,thread))*Diameter_01/Viscosity_01,-0.198);
    else
        Cx=1.156*pow(Density_01*fabs(C_U(cell,thread))*Diameter_01/Viscosity_01,-0.2647);
    source=Coefficient_01*Cx*fabs(C_U(cell,thread))*C_U(cell,thread);
    dS[eqn]=2.0*Coefficient_01*Cx*fabs(C_U(cell,thread));
    return source;
}

DEFINE_SOURCE(y_momentum_source,cell,thread,dS,eqn)
{
    real source;
    if(Density_01*fabs(C_V(cell,thread))*Diameter_01/Viscosity_01<8000)
        Cy=0.619*pow(Density_01*fabs(C_V(cell,thread))*Diameter_01/Viscosity_01,-0.198);
    else
        Cy=1.156*pow(Density_01*fabs(C_V(cell,thread))*Diameter_01/Viscosity_01,-0.2647);
source = Coefficient_01 * Cy * fabs(C_V(cell,thread)) * C_V(cell,thread);

dS[eqn] = 2.0 * Coefficient_01 * Cx * fabs(C_V(cell,thread));

return source;
}

DEFINE_SOURCE(z_momentum_source, cell, thread, dS, eqn)
{
real source;

if(Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01 >= 2250 && Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01 < 25000)
    Cz = -0.131 * pow(Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01, -0.294);
else if(Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01 >= 25000)
    Cz = -0.066 * pow(Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01, -0.227);
else
    Cz = 31.0 / (Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01);

source = Coefficient_02 * Cz * fabs(C_W(cell,thread)) * C_W(cell,thread);

dS[eqn] = -2.0 * Coefficient_02 * Cz * fabs(C_W(cell,thread));

return source;
}

DEFINE_SOURCE(turbulent_kinetic_energy_source, cell, thread, dS, eqn)
{
real source;

if(Density_01 * fabs(C_U(cell,thread)) * Diameter_01 / Viscosity_01 < 8000)
    Cx = 0.619 * pow(Density_01 * fabs(C_U(cell,thread)) * Diameter_01 / Viscosity_01, -0.198);
else
    Cx = 1.156 * pow(Density_01 * fabs(C_U(cell,thread)) * Diameter_01 / Viscosity_01, -0.2647);

if(Density_01 * fabs(C_V(cell,thread)) * Diameter_01 / Viscosity_01 < 8000)
    Cy = 0.619 * pow(Density_01 * fabs(C_V(cell,thread)) * Diameter_01 / Viscosity_01, -0.198);
else /* Rey >= 8000 */
    Cy = 1.156 * pow(Density_01 * fabs(C_V(cell,thread)) * Diameter_01 / Viscosity_01, -0.2647);

if(Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01 >= 2250 && Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01 < 25000)
    Cz = -0.131 * pow(Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01, -0.294);
else if(Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01 >= 25000)
    Cz = -0.066 * pow(Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01, -0.227);
else
    Cz = 31.0 / (Density_01 * fabs(C_W(cell,thread)) * Diameter_01 / Viscosity_01);

source = Coefficient_02 * Cz * fabs(C_W(cell,thread)) * C_W(cell,thread);

dS[eqn] = -2.0 * Coefficient_02 * Cz * fabs(C_W(cell,thread));

return source;
l(thread) * Diameter_01 / Viscosity_01 < 25000)

Cz = 0.131 * pow(Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01, 0.294);
else if (Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01 >= 25000)

Cz = 0.066 * pow(Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01, 0.227);
else

Cz = 31.0 / (Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01);

source = 0.5 * Coefficient_01 * Cx * pow(fabs(C_U(cell, thread)), 3.0) + 0.5 * Coefficient_01 * Cy * pow(fabs(C_V(cell, thread)), 3.0) + 0.5 * Coefficient_02 * Cz * pow(fabs(C_W(cell, thread)), 3.0);

dS[eqn] = 0;

return source;

}

DEFINE_SOURCE(turbulent_dissipation_rate_source, cell, thread, dS, eqn)

{ real source;

if (Density_01 * fabs(C_U(cell, thread)) * Diameter_01 / Viscosity_01 < 8000)

Cx = 0.619 * pow(Density_01 * fabs(C_U(cell, thread)) * Diameter_01 / Viscosity_01, 0.198);
else

Cx = 1.156 * pow(Density_01 * fabs(C_U(cell, thread)) * Diameter_01 / Viscosity_01, 0.2647);

if (Density_01 * fabs(C_V(cell, thread)) * Diameter_01 / Viscosity_01 < 8000)

Cy = 0.619 * pow(Density_01 * fabs(C_V(cell, thread)) * Diameter_01 / Viscosity_01, 0.198);
else

Cy = 1.156 * pow(Density_01 * fabs(C_V(cell, thread)) * Diameter_01 / Viscosity_01, 0.2647);

if (Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01 >= 2250 && Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01 < 25000)

Cz = 0.131 * pow(Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01, 0.294);
else if (Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01 >= 25000)

Cz = 0.066 * pow(Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01, 0.227);
else

Cz = 31.0 / (Density_01 * fabs(C_W(cell, thread)) * Diameter_01 / Viscosity_01);
References


