Title
Determination of the Number of Tube Rows to Obtain Closure for Volume Averaging Theory Based Model of Fin-and-Tube Heat Exchangers

Permalink
https://escholarship.org/uc/item/9299x14g

Journal
Journal of Heat Transfer, 133(12)

ISSN
0022-1481

Authors
Zhou, Feng
Hansen, Nicholas E
Geb, David J
et al.

Publication Date
2011

DOI
10.1115/1.4004478

Peer reviewed
Determination of the Number of Tube Rows to Obtain Closure for Volume Averaging Theory Based Model of Fin-and-Tube Heat Exchangers

Modeling of fin-and-tube heat exchangers based on the volume averaging theory (VAT) requires proper closure of the VAT based governing equations. Closure can be obtained from reasonable lower scale solutions of a computational fluid dynamics (CFD) code, which is the tube row number chosen should be large enough, so that the closure can be evaluated for a representative elementary volume (REV) that is, not affected by the entrance or recirculation at the outlet of the fin gap. To determine the number of tube rows, three-dimensional numerical simulations for plate fin-and-tube heat exchangers were performed, with the Reynolds number varying from 500 to 6000 and the number of tube rows varying from 1 to 9. A clear perspective of the variations of both overall and local friction factor and the Nusselt number as the tube row number increases are presented. These variation trends are explained from the viewpoint of the field synergy principle (FSP). Our investigation shows that 4 + 1 + 1 tube rows is the minimum number to get reasonable lower scale solutions. A computational domain including 5 + 2 + 2 tube rows is recommended, so that the closure formulas for drag resistance coefficient and heat transfer coefficient could be evaluated for the sixth and seventh elementary volumes to close the VAT based model. (DOI: 10.1115/1.4004478)

Keywords: volume averaging theory, closure, field synergy principle, fin-and-tube heat exchanger, representative elementary volume

Introduction

Volume averaging theory (VAT) is an approach that can be applied to many different types of transport phenomena [1–7] and has been used to rigorously cast the point-wise conservation of energy, momentum and mass equations into a form that represents the thermal and hydraulic properties of fin-and-tube heat exchanger morphology. Using the VAT to optimize the flow and heat transfer in heat exchangers, a more general model that can easily be adapted to many different structures can be made. By modeling heat exchangers as porous media, specific geometry can be accounted for in such a way that the details of the original structure can be replaced by their averaged counterparts and the governing VAT equations can be solved for a wide range of heat exchanger designs. The tricky part about using VAT is that proper closure is needed to complete the governing equations. Closure for both the VAT based momentum and energy equations have been strictly deduced and defined by Travkin and Catton [3], making it possible to use lower scale flow and temperature fields to obtain closure for VAT based governing equations of heat exchangers.

The closure formula for momentum equation is a drag coefficient represented by

\[
\begin{align*}
\mathbf{c}_d &= 2 \int_{S_w} \frac{\rho \cdot d\vec{s}}{\rho \bar{u}^2 A_w} \left( \frac{\partial}{\partial x} \left( \frac{\rho \bar{u}}{\bar{u}^2} \right)_f \right) + 2 \int_{S_w} \frac{\rho \cdot d\vec{s}}{\rho \bar{u}^2 A_w} \left( \frac{\partial}{\partial y} \left( \frac{\rho \bar{u}}{\bar{u}^2} \right)_f \right) \\
&= \frac{1}{2 \rho \bar{u}^2} + \frac{\partial}{\partial x} \left( \frac{\rho \bar{u}}{\bar{u}^2} \right)_f + \frac{\partial}{\partial y} \left( \frac{\rho \bar{u}}{\bar{u}^2} \right)_f
\end{align*}
\]

The closure formula for energy equation is the heat transfer between phases given by

\[
h = \frac{1}{\Delta \Omega} \int_{S_w} (k_f + k_v) \nabla T_f \cdot d\vec{s} - \rho c_p \nabla \left( \frac{m c_p}{\Delta \Omega} \frac{T_f}{\Delta \Omega} \right) + \frac{\rho c_p \nabla \left( \frac{m c_p}{\Delta \Omega} \frac{T_f}{\Delta \Omega} \right)}{S_w(T_f - T_j)} + \frac{\rho c_p \nabla \left( \frac{m c_p}{\Delta \Omega} \frac{T_f}{\Delta \Omega} \right)}{S_w(T_f - T_j)}
\]

The closure form can be obtained by applying Eqs. (1) and (2) on a selected representative elementary volume (REV) of particular heat exchanger morphology. To treat a fin-and-tube heat exchanger as porous media, entrance effects are important for both momentum and energy transport. When flow enters porous media, temperature and velocity profiles are uniform across the entrance area and as the flow progresses further downstream from the inlet, both thermal and momentum boundary layer are getting thicker until they merge with its counterpart at some particular location. Also, local values differ from the overall averaged as expected, therefore special attention should be paid when one uses the drag coefficient and heat transfer coefficient with the VAT governing equations to solve large scale problem. Local values are the only values that have a physical meaning when describing transport phenomena with the VAT macro scale equations. Therefore, before evaluating the friction factor \( \mathbf{c}_d \) and heat transfer coefficient \( h \) from the output of a CFD code, the tube row number one needs to simulate to obtain reasonable lower scale solutions must be determined.

Extensive investigations on the performance of fin-and-tube heat exchanger have been done, either experimentally or numerically. Rich [8] presented the experimental results for six coils,

Contributed by the Heat Transfer Division of ASME for publication in the Journal of Heat Transfer. Manuscript received January 3, 2011; final manuscript received June 19, 2011; published online October 6, 2011. Assoc. Editor: Sujay Kumar Saha.
with the number of tube rows in the direction of air flow varying from 1 to 6. It was concluded that the pressure drop per row is independent of the number of tube rows. McQuiston [9] proposed the first well-known correlation for plate fin-and-tube heat exchangers with tube row number being in the range of 1–4. Based on a superposition model which was initially proposed by Rich [10], Gray and Webb [11] gave an updated correlation for fin-and-tube heat exchangers that is superior to McQuiston’s [9]. It should be noted that the correlations were based on the experimental data for 4-row fin-and-tube heat exchangers. Kang et al. [12] presented experimental data and correlations for a 3-row fin-and-tube heat exchanger core in a wide range of the Reynolds number. Most recently, Wang et al. [13] proposed what is at this time the most precise correlations for the friction factor and heat transfer coefficient on the air side of fin-and-tube heat exchangers. The correlations are based on a total of 74 samples and the proposed heat transfer correlation can describe 88.6% of the database within ±15%, while the proposed friction correlation can correlate 85.1% of the database within ±15%. However, it is applicable only for up to six tube rows. Besides experimental investigations, many numerical simulations of plate fin-and-tube heat exchangers were reported. Torikoshi et al. [14] reported simulation results for 1-row and 2-row fin-and-tube heat exchangers. Jang et al. [15–17] did extensive numerical and experimental studies on fluid flow and heat transfer over a multirow (1–6 rows) fin-and-tube heat exchanger. It was reported that the number of tube rows has a small effect on the average heat transfer coefficient when the row number was larger than 4.

After reviewing the literature, there are three problems we need to point out. The first is that most of the studies reviewed above were limited to small number of tube rows. This is not enough to determine how large the tube row number must be to simulate the process in order to get reasonable lower scale solutions. The second that should be noted is that most of these investigations are based on the average parameters over the whole domain. While for VAT based modeling, only the local values can be used in calculations to accurately predict total heat transfer in porous media. Therefore, the influence of tube row number on local heat transfer characteristics needs to be clearly revealed. The third problem is related to the air side recirculation between the fin-and-tube region and the extended region. The elementary volumes being affected by the recirculation should not be used to calculate the closure for the VAT based model. The above three issues are the motivation of the present study.

In the current paper, 3D numerical simulations for plate fin-and-tube heat exchangers were performed, with the Reynolds number varying from 500 to 6000 and tube row number varying from 1 to 9. A clear perspective of the variations of both overall and local friction factor and Nusselt number as the tube row number increases was presented. These variation trends are also explained from the view point of field synergy principle, a novel concept proposed and verified by Guo and some other researchers [18–22]. The basic mechanism of FSP is that the convective heat transfer could be enhanced by decreasing the intersection angle between the velocity and the temperature gradient. The changing tendency of this intersection angle, in turn, provides a helpful means to explain the variation in trends of the heat transfer coefficient and could help to further verify the tube row number one should simulate to get reasonable lower scale solutions for the VAT based modeling. Considering that both the traditional analysis method and the novel FSP analysis came to the same conclusion, a specific tube row number is proposed.

**Physical Model and Numerical Method**

**Physical Model.** A schematic diagram of a plate fin-and-tube heat exchanger is shown in Fig. 1. There usually three or more rows of tubes which are arranged in-line or staggered. The dimensions of the main geometrical parameters, like tube diameter ($D_t$, $D_o$, and $D_c$), tube transverse pitch ($P_t$), tube longitudinal pitch ($P_l$), fin pitch ($P_f$), and fin thickness ($\delta_f$) are tabulated in Table 1, which are the same as those of the sample core tested by Kang et al. [12] experimentally.

Here we consider the hot air flow across the fin side, while the cooling water flows through the tubes. The tubes and fins are constructed of copper and aluminum, respectively. Both fluids are assumed to be incompressible with constant properties. Due to the relatively large heat transfer coefficient on the tube side, the tube inner wall temperature was set equal to the fluid temperature. The conjugate effect of the tube wall was treated enabling the fin effect to be properly incorporated into the problem [23].

**Flow Model and Governing Equations.** In this study, laminar steady model is adopted for the flow within the Reynolds number range of 500–6000. The choice of flow model is explained as follows. First, if the flow between two adjacent fins is considered as channel flow, the transition Reynolds number is around 3200, with the double fin spacing as the reference length for Re. In this study, the fin collar diameter is 10.55mm, which is about 5.3 times of the fin spacing, thus the transition Reynolds number of 2300 corresponds to around 6066 based on the fin collar diameter. If the flow between two adjacent fins is regarded as the flow past a cylinder, the turbulent flow occurs when the Reynolds number is over $1.4 \times 10^5$ [24]. Second, although flow visualization shows that when the Reynolds number is over 100, unsteady wake occurs behind the cylinder, from engineering point of view, we care more about the averaged parameters, like averaged friction factor and the Nusselt number. Third, many studies, which adopted steady and laminar model for flow and heat transfer analysis of fin-and-tube heat exchangers, obtained reasonably good results and have been published in international journals [15–17,25,26]. Finally, to further verify the laminar assumption, both laminar model and SST model were adopted to calculate the case for Reynolds number of 6000, the difference turned out to be negligible.

**Table 1 Some data of the fin-and-tube heat exchanger**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tube row arrangement</td>
<td>Staggered</td>
</tr>
<tr>
<td>Tube material</td>
<td>Copper</td>
</tr>
<tr>
<td>Fin material</td>
<td>Aluminum</td>
</tr>
<tr>
<td>Tube inner diameter, $D_c$</td>
<td>9.33 mm</td>
</tr>
<tr>
<td>Tube outer diameter, $D_o$</td>
<td>10.15 mm</td>
</tr>
<tr>
<td>Fin collar diameter, $D_e$</td>
<td>10.55 mm</td>
</tr>
<tr>
<td>Tube transverse pitch, $P_t$</td>
<td>25 mm</td>
</tr>
<tr>
<td>Tube longitudinal pitch, $P_l$</td>
<td>21.65 mm</td>
</tr>
<tr>
<td>Fin thickness, $\delta_f$</td>
<td>0.2 mm</td>
</tr>
<tr>
<td>Fin pitch, $P_f$</td>
<td>2.2 mm</td>
</tr>
</tbody>
</table>
The governing equations for continuity, momentum and energy in the computational domain can be expressed as follows:

Continuity equation

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (3)$$

Momentum equation

$$\frac{\partial}{\partial x_i} (u_i u_j) = \frac{\mu}{\rho} \frac{\partial u_i}{\partial x_j} - \frac{1}{\rho} \frac{\partial p}{\partial x_i} \quad (4)$$

Energy equation

$$\frac{\partial (u_i T)}{\partial x_i} = \frac{k}{\rho c_p} \frac{\partial T}{\partial x_i} \quad (5)$$

The boundary conditions assigned for the computational domain are tabulated in Table 2.

SC/Tetra HPC Solver was used to solve the governing equations. The numerical techniques and discretization schemes used in the code are a 2nd order Monotone Upstream-centered Schemes for Conservation Laws for convective terms in the energy and momentum equations while for the momentum and energy diffusion terms, an accuracy weighted scheme is used. A corrected version of the Semi-Implicit Method of the Pressure Linked Equations (SIMPLEC) is used [23, 27] for the pressure correction.

Computational Domain and Grid System. Since the fin-and-tube heat exchanger has symmetry and periodicity in the y and z directions, the cell between two adjacent fin surfaces is simulated with half of the fin thickness at the upper and the bottom sides, just as Figs. 1(b)–1(d) shows. The actual length of computation domain was 7.5 times the fin and tube region in the flow direction, see Fig. 2. Because of the thickness of the fin, the air velocity profile at the entrance is not uniform. The computational domain is then extended upstream 1.5 times the stream-wise fin length, so that a uniform velocity distribution can be ensured at the domain inlet. The computational domain is extended downstream 5 times the stream-wise fin length, so that at the outer flow boundary no flow recirculation exits and the local one-way method can be used for the numerical treatment of the outer flow boundary condition [27].

A commercial finite volume code SC/Tetra v8 (2009) is used to do the 3D CFD simulation. This code utilizes a vertex based scheme for which the discrete equations are written in a relatively simple form and the number of control volumes is reduced compared to cell centered formulation. The grid system for the configurations is built by SC/Tetra Pre, which controls the size of the mesh three dimensionally by creating an octree. When creating an octree, octants are refined automatically to prevent them from differing by two or more levels from the next octant. Mesh adaptive analysis was used to build the grid system. Fine elements are arranged where the change of pressure, velocity or temperature is little change occurs, especially in the extended region. It can also be observed that the grid system has gradual variation from fine elements to coarse elements, which is quite important to maintain the calculation accuracy.

Grid independence tests were made carefully, and the numerical results obtained can be regarded as grid-independent. Take the two-row model as an example, where grid systems with 6,978,849, 1,728,574, 2,639,059, 3,860,677, 4,985,479, 6,607,223, and 8,177,712 elements were tested, and comparing the results of the finest grid with the second finest one, a 1.3% difference was yielded. To save computer resources but also get a reasonable accuracy, the sixth grid system was adopted. Table 3 shows the element numbers used for the nine cases of our computation.

### Table 2 Boundary conditions

<table>
<thead>
<tr>
<th>Inlet</th>
<th>$u = \text{const}, \nu = \nu = \text{const}, T = \text{const}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet</td>
<td>$\frac{\partial u_i}{\partial x_i} = \frac{\partial T}{\partial x_i} = 0$</td>
</tr>
<tr>
<td>Eight surfaces of the extended region</td>
<td>symmetric, slip, and adiabatic wall</td>
</tr>
<tr>
<td>Interface between air and solid</td>
<td>no-slip, no thermal resistance</td>
</tr>
<tr>
<td>Tube inside wall</td>
<td>$u = \nu = \text{const}, T_e = \text{const}$</td>
</tr>
<tr>
<td>The other surfaces</td>
<td>symmetric</td>
</tr>
</tbody>
</table>

### Parameter Definitions

Definitions for the characteristic quantities which will be used in the analysis of numerical results are presented the following:

$$Re = \frac{\rho \phi_{\text{max}} D_{ij}}{\mu} \quad (6)$$

$$Nu = \frac{h D_i}{k} \quad (7)$$

$$h = \frac{\Phi}{A_o \Delta T \eta_o} \quad (8)$$

$$\Phi = \rho c_p (T_{\text{in}} - T_{\text{out}}) \quad (9)$$

$$\Delta T = \frac{T_{\text{max}} - T_{\text{min}}}{\log(T_{\text{max}}/T_{\text{min}})} \quad (10)$$

$$\eta_o = 1 - \frac{A_f}{A_o} (1 - \eta_f) \quad (11)$$

$$A_f = N \left[ P \phi_1 - \frac{D_k}{2} \right]^2 \quad (12)$$

$$A_o = N \left[ P \phi_1 - \frac{D_k}{2} \right]^2 + \frac{1}{2} \pi D_i (F_p - \delta_f)$$

![Fig. 2 Computational domain (It is not drawn to scale)](image)

![Fig. 3 Grid system for 2-row case](image)
where $\eta_{\text{max}}$ is the mean velocity at the minimum cross section, $T_{\text{max}} = \max(T_{\text{in}} - T_{\text{w}}, T_{\text{out}} - T_{\text{w}})$, $T_{\text{mid}} = \min(T_{\text{in}} - T_{\text{w}}, T_{\text{out}} - T_{\text{w}})$, $\Phi$ is the actual heat transfer rate between air and the fin surface.

As has been discussed above, the computation is of a conjugated type where the fin efficiency is determined during the computations and cannot be obtained in advance. According to heat transfer theory [28,29], the fin efficiency is defined as the actual heat transfer rate from the fin and tube when the fin is at the same temperature as the tube. In our numerical simulations, the fin efficiency $\eta_f$ is computed by the approximation method described by Schmidt [30] and is as follows:

$$\eta_f = \frac{\tanh(mr\phi)}{mr\phi}$$

(16)

where

$$m = \sqrt{\frac{2h}{k_f d_f}}$$

(17)

$$\phi = \left(\frac{R_{eq}}{r} - 1\right) \left[1 + 0.35 \ln \left(\frac{R_{eq}}{r}\right)\right]$$

(18)

For a staggered tube layout

$$\frac{R_{eq}}{r} = 1.27 \frac{X_M}{r} \left(\frac{X_L}{X_M} - 0.3\right)^{1/2}$$

(19)

$$X_L = \sqrt{\frac{(P_r/2)^2 + P_r^2}{2}}$$

(20)

$$X_M = P_r/2$$

(21)

For an inline tube layout, or a 1-row fin-and-tube heat exchanger

$$\frac{R_{eq}}{r} = 1.28 \frac{X_M}{r} \left(\frac{X_L}{X_M} - 0.2\right)^{1/2}$$

(22)

$$X_L = P_r/2$$

(23)

$$X_M = P_r/2$$

(24)

To analyze the numerical results using the field synergy principle, the concept of FSP is briefly reviewed as follows.

For a steady-state elliptic flow of constant properties within an computational domain (taking the fluid flow and heat transfer over a backward step [31] as an example, as shown in Fig. 4), its energy conservation equation can be described as

$$\rho c_p \left[ u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right] = k \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right)$$

(25)

Integrate Eq. (25) over the entire domain $\Omega_{\text{abcde}}$, leading to

$$\int_{\Omega_{\text{abcde}}} \rho c_p (\vec{U} \cdot \nabla T) \, dA = \int_{\Omega_{\text{abcde}}} k \nabla^2 T \, dA$$

(26)

Incorporating the Gauss law for reduction of the integral dimension, the right side of Eq. (26) can be written as

$$\int_{\Omega_{\text{abcde}}} k \nabla^2 T \, dA = \int_{\text{abc}} k f \vec{n} \cdot \nabla T \, ds + \int_{\text{de}} k f \vec{n} \cdot \nabla T \, ds + \int_{\text{cd}} k f \vec{n} \cdot \nabla T \, ds$$

(27)

The last two terms on the right-hand side of Eq. (27) represent the axial heat conduction within the fluid while the first two terms stand for the convective heat transfer between the fluid and solid. For flow with Peclet number greater than 100, the axial conduction terms can be neglected [32], leading to

$$\int_{\Omega_{\text{abcde}}} \rho c_p (\vec{U} \cdot \nabla T) \, dA = \int_{\text{abc}} k f \vec{n} \cdot \nabla T \, ds + \int_{\text{de}} k f \vec{n} \cdot \nabla T \, ds$$

(28)

For conventional working fluids adopted in engineering, the Peclet numbers are usually greater than 100, and hence, the integration $\int_{\Omega_{\text{abcde}}} \rho c_p (\vec{U} \cdot \nabla T) \, dA$ actually represents the energy transferred by convection [31]. The inner production $\vec{U} \cdot \nabla T = |\vec{U}| |\nabla T| \cos \theta$, in which $\theta$ is the local intersection angle between the local velocity vector and the temperature gradient. Therefore, it is obvious that besides increasing the velocity and temperature gradient, decreasing their intersection angle $\theta$ will also make the integration $\int_{\Omega_{\text{abcde}}} \rho c_p (\vec{U} \cdot \nabla T) \, dA$ larger, enhancing the heat transfer. On the other hand, for a given flow rate and temperature gradient, the changing tendency of this intersection angle helps explain the variation in trends of the convective heat transfer coefficient, which further verifies the tube row number one should simulate to get reasonable lower scale solutions for the VAT based modeling.

According to the concept of this principle, the following quantity is introduced

$$M = \sum_{n} \left(\frac{|\vec{U}| |\nabla T|}{n}\right)$$

(29)

The local intersection angle between the velocity and temperature gradient is

$$\theta = \cos^{-1} \left(\frac{\frac{\partial T}{\partial x} + \frac{\partial T}{\partial y} + \frac{\partial T}{\partial z}}{|\vec{U}| |\nabla T|}\right)$$

(30)
Numerical Results and Discussion

Validation of the CFD Code and the Simulation Method. To verify the computational model and the method adopted in numerical simulation, preliminary computations were first conducted for a 3-row plate fin-and-tube heat exchanger. The dimensions used in the simulation are the same as those of the heat exchanger measured by Kang et al. [12] for his experiments.

The Nusselt number and friction factor obtained from simulation solutions were compared with the correlations and experimental data by Kang et al. [12] and the correlations by Wang et al. [13] and are shown in Fig. 5. From Fig. 5 we can see that the maximum deviation of the friction factor and the Nusselt number from experiment are 10.4% and 11.9% with the average deviation being around 6% and 5.5%, respectively. Our predicted results and the experimental data agree very well, thereby showing the reliability of the physical model and the adopted numerical method.

Validation of FSP Application. To verify that the FSP is suitable to our problem, computation was conducted for the 3-row case and the effect of Reynolds number on heat transfer characteristic of plate fin-and-tube heat exchangers was analyzed by FSP. The Reynolds number based on the fin collar outside diameter varied from 500 to 6000 and the corresponding air frontal velocity was ranged from 0.38 m/s to 4.6 m/s.

Figure 6 shows the variation of $M/M_0$ as a function of the Reynolds number, where $M_0$ is the value of $M$ when $Re = 500$. In the Reynolds number range of our computation, the increase in air flow velocity is the main reason why the heat transfer rate increased. Therefore, it is not surprising that the value of $M$ has a nearly linear-increasing relationship with the Reynolds number, which should lead the Nusselt number to be proportional to $Re$. However, the increasing trend of the average Nusselt number with $Re$ was weakened as the Reynolds number increases, just as Fig. 7 shows. The reason why this happened is that the intersection angle between velocity and temperature gradient increased with $Re$, shown in Fig. 7, which leads to the worsening of the synergy.

Consequently, the increasing trend of the intersection angle is similar to that of the Nusselt number.

Figures 8–10 present the local distributions of velocity (a), temperature (b) and intersection angle (c) on the x-y plane at $z = 0.0011$ m for $Re = 500, 3000$ and 6000, respectively. Basically, these pictures reveal two main pieces of information:

1. The synergy becomes worse in the direction of air flow stream. At the inlet region, the isothermals are almost perpendicular to the velocity vector, which leads to a quite small intersection angle at the inlet part. After the flow passes through the first tube, the velocity vector almost parallels the isothermals, which results in a worse synergy.

2. That synergy becomes worse with increasing the Reynolds number can be observed qualitatively by noting region of red color in the paint graphs of intersection angle distribution. Comparing Fig. 10 with Fig. 8, we find that for large Reynolds number, even at the inlet, the synergy is not good, let alone the remaining part of the simulation region. Therefore, the increase of heat transfer rate brought by the increase of air flow velocity is counteracted partly by the deteriorated synergy.

Based on the above analysis, the increasing trend of Nusselt number with Reynolds number was explained well by the FSP. Thus, it is meaningful to use FSP to analyze the heat transfer characteristic of plate fin-and-tube heat exchangers and help to determine the tube row number for our purpose without any simplifying assumption, such as ideal fin and ideal temperature difference, which was adopted by studies [19–22] while verifying the FSP.

Determine the Length of Computational Domain. It should be noted that the number of tube rows has a different effects on the friction factor than the Nusselt number of fin-and-tube heat exchangers. Fig. 11, which was plotted using the correlations given by Wang et al. [13] illustrates the difference.

As can be seen in Fig. 11, for fin-and-tube heat exchangers with multiple-row tubes, the friction factors are almost independent of the number of tube rows, while the Nusselt number decreases with the increasing of tube row number. Our simulation results, conducted with the tube row number varying from 1 to 9, came to the same conclusion, shown by Fig. 12. It can be seen that the variation of friction factor with $N$ is quite small, and when the row number is larger than 3, $f$ could be considered independent of $N$. Wang et al. [33,34] also arrived at the same conclusion by means of experiments. It is also shown that the Nusselt number decreases with the increasing of tube row number $N$, and when $N > 4$, the decreasing trend slows down. When $N > 6$, the variations of

\[
\theta_m = \sum_{i,j,k} \frac{dV_{ijk}}{dV_{ijk}}
\]

where $dV_{ijk}$ is the volume element of the control volume $(i,j,k)$.

![Fig. 5 Comparison between the present CFD results and well-known correlations](image1)

![Fig. 6 Variation curve of $M/M_0$ with Re](image2)
friction factor and Nusselt number are quite subtle, making it reasonable to conclude that the effect of tube row on the heat transfer and fluid flow could be neglected when the number of tube rows is greater than six.

Figure 13 shows the variation of $M/M_0$ and $\theta_m$ with the increasing tube row number, where $M_0$ is the $M$ value for 1-row case. As can be seen, the average product of $|U|$ and $|\text{grad}T|$ decreases with the increasing of $N$. It also can be found that the intersection angle between air flow velocity and temperature gradient increases with the increasing of tube row number and the increasing trend slows down when $N > 4$. This explains well the change in trend of Nusselt with the increasing of $N$.

To get a more vivid impression, two more figures, Figs. 14 and 15, together with Fig. 9, comparisons of local distributions of velocity vector, isothermals and intersection angle on the x-y plane for 3-row case, 6-row case, and 9-row case are presented. It is obvious that as the number of tube rows increases, the percentage of the area which has deteriorated synergy (red area) increases. It is also found that when $N > 6$, the flow field, temperature field and the distribution of intersection angle are marked with obvious periodicity, shown by the dashed frame in Fig. 15.

It should be noted that the above discussion is based on values averaged over the whole simulation domain, while to use volume average theory to optimize the heat exchangers, we are more concerned about values averaged over a selected REV. As a result, although we arrived at the conclusion that when the number of tube rows is larger than six, the flow field and thermal field could be considered as periodic in the streamwise direction, we cannot say that the computational domain including six tube rows is long enough to get a reasonable lower scale solution. In-depth observations should be carried out on the variation of concerned parameters for every single REV along the air flow direction. Thus, we divided the computational domain of the 7-, 8-, and 9-row cases into 7, 8, and 9 REVs and then calculated the Nusselt numbers, $M$ values and intersection angles for the total 24 elementary volumes, and plotted the variation curves in Fig. 16. Some conclusions that can be drawn from the graphs are the following:

1. The averaged local $M$ value over every single elementary volume decreases along with the direction of air flow. This is why the magnitude of $M$ averaged over the whole domain decreases as the number of tube rows increase.
2. Since the curves of $M$ for the three cases collapse to a single curve, it is reasonable to say that the number of tube rows has no effect on the trend of local $M$ from the domain inlet all the way to the outlet.
(3) Excluding the last data point, the curves of $\text{Nu}$ and $\theta_m$ also collapse to two single curves. However, the $\text{Nu}$ number and intersection angle for the last elementary volume is unusual due to the reason that the air-flow recirculation happens when it flows from the trailing edge of the fin to the extended domain. For this reason, the solution of the last volume shouldn’t be used to calculate the closure for VAT based model.

Therefore, in conclusion, $N > 4$ is needed to get a believable local value, which means that the tube row number should not be...
less than $4 + 1 + 1$ rows (the first 4 rows is to develop the velocity and temperature field, and the fifth elementary volume is where to calculate the closure. The solution of the 6th row is not for use considering the invalidity of the last data point). We recommend that $5 + 2 + 2$ rows be simulated and that the closure be obtained by averaging the local value over the sixth and seventh elementary volumes.

Concluding Remarks

The present paper describes an effort to determine the tube row number that will be large enough to obtain good estimates of the closure parameters for the VAT-based modeling of fin-and-tube heat exchangers. Simulations were conducted from 1-row to 9-row heat exchangers. The variations of overall Nusselt number and friction factor as well as local values with the increasing of row number were presented. These variation trends were also explained from the viewpoint of the FSP. It was found that when $N > 3$, the friction factor could be considered independent of $N$. As to the Nusselt number, when $N > 4$, the variation trend of $N_u$ tends to ease up, which agrees well with the increasing trend of intersection angle. When $N > 6$, the variations of both local averaged Nusselt number and intersection angle are subtle, and can be regarded as independent of $N$. For this reason, $4 + 1 + 1$ rows of fin-and-tube heat exchanger is the smallest number that will yield a believable local value and a computational domain with $5 + 2 + 2$ tube rows is recommended. The present paper brings us step closer to obtaining closure for the CFD based modeling of fin-and-tube heat exchanger from CFD solutions, which is the subject of future work.

Acknowledgment

The support of a Department of Energy NERI grant, Award Number DE-FG07-07ID14827, is gratefully acknowledged. The numerical calculations were performed using commercial CFD software SC/Tetra Version 8, a product of Software Cradle Co., Ltd.

Nomenclature

$A = \text{area, m}^2$

$A_f = \text{fin surface area, m}^2$

$A_o = \text{total surface area, m}^2$

$A_{wp} = \text{the cross flow projected area, m}^2$

$A_w = \text{wetted surface, m}^2$

$c_d = \text{drag coefficient}$

$c_p = \text{specific heat, J/(kg \cdot K)}$

$D_o = \text{outer diameter of the tube, m}$

$D_i = \text{inner diameter of the tube, m}$

$D_e = \text{fin collar outside diameter, } D_e = D_o + 2\delta_f, \text{ m}$

$D_b = \text{hydraulic diameter, m}$

$F_p = \text{fin pitch, m}$

$f = \text{Moody friction factor}$

$h = \text{heat transfer coefficient, W/(m}^2 \cdot \text{K)}$

$k = \text{thermal conductivity, W/(m} \cdot \text{K)}$

$m = \sqrt{\frac{2h}{k \delta_f}}$, parameter

$(m) = \text{the average porosity}$

$m = \text{mass flow rate, kg/s}$

$n = \text{the number of control volumes}$

$M = \sum (U \nabla U)$, parameter

$M_0 = \text{the base value of, } M$

$N = \text{the number of tube rows}$

$Nu = \text{Nusselt number}$

$p = \text{pressure, Pa}$

$\Delta p = \text{pressure drop, Pa}$

$Pr = \text{Prandtl number}$

$P_t = \text{transverse tube pitch, m}$

$P_l = \text{longitudinal tube pitch, m}$

$r = \text{radius of tube, including collar thickness, m}$

$Re = \text{Reynolds number scaled with fin collar outside diameter and maximum velocity, } Re = \frac{2u_{max}}{v}$

$Re_q = \text{equivalent radius for circular fin, m}$

$S_w = \text{specific surface, 1/m}$

$S_{wp} = \text{the cross flow projected area per volume, 1/m}$

$T = \text{fluid temperature, K}$

$\Delta T = \text{logarithmic mean temperature difference, K}$

$T_s = \text{solid temperature, K}$

$u, v, w = x-, y-, z-$direction velocity term, m/s

$U = \text{velocity vector}$

$X_L = \text{geometric parameter, m}$

$X_M = P_t/2$, geometric parameter, m

Greek Symbols

$\delta_f = \text{thickness of a fin, m}$

$\eta_s = \text{surface efficiency}$

$\eta_f = \text{fin efficiency}$

$\Theta = \text{intersection angle, } ^\circ$

$\mu = \text{viscosity, Pa} \cdot \text{s}$

$\nu = \text{kinetic viscosity, m}^2/\text{s}$

$\rho = \text{density, kg/m}^3$

$\tau_{nl} = \text{laminar shear stress, Pa}$

$\tau_{tu} = \text{turbulent shear stress, Pa}$

$\Phi = \text{heat transfer rate, W}$

$\Delta \Omega = \text{volume of the REV, m}^3$

Subscripts and Superscripts

$\sim = \text{a value in the fluid averaged over the representative volume}$

$\bar{} = \text{indicates an average of turbulent values}$

$\wedge = \text{fluctuation of a value}$

$(f)_T = \text{means the superficial average of the function, } f$

$f = \text{fin surface or fluid phase}$

$\text{in = air-side inlet}$

$\text{max = maximum value}$

$m = \text{mean value}$

$\text{out = air-side outlet}$

$s = \text{solid phase}$

$T = \text{turbulent}$

$w = \text{tube wall}$

References


