Numerical Analysis of a Multi-Row Multi-Column Compact Heat Exchanger

Ashkan Motamedi¹, Arturo Pacheco-Vega¹*, J. Rafael Pacheco²

¹Department of Mechanical Engineering, California State University, Los Angeles, Los Angeles, CA 90032, USA
²School of Mathematical and Statistical Sciences, Arizona State University, Tempe, AZ 85287, USA
E-mail: *apacheco@calstatela.edu

Abstract. In the present study we carry out three-dimensional fluid flow and heat transfer simulations on the external side of a compact heat exchanger to analyze the interaction between the fluid and its geometry. The overall objective is to use the resulting information for the design of more compact devices. The type of heat exchanger considered here is the common plain-fin and tube, with air flowing over the tubes and water as the inner-tube fluid. Two heat exchanger configurations, in which the tube arrangement is either in-line or staggered, conform the basic geometries. The size of the heat exchanger –regardless of the type of arrangement– which serves as the baseline for the parametric analysis, is defined by fixing its length; i.e., the number of rows in the flow direction. For the two heat exchanger configurations examined here, the dimensional form of the governing equations, along with the corresponding boundary conditions, are solved under specific flow and temperature values using a finite element method to compute the velocity, pressure and temperature fields. From these, the heat transfer rate and pressure drop are then calculated. The computations are performed for a range in the values of the Reynolds number within the laminar regime. For all cases considered, results from this investigation indicate that the geometrical arrangement plays a major role in the amount of heat being exchanged and that, for a given device, the length needed to exchange 99% of the corresponding amount of energy that may be transferred by the baseline model, is confined to less than 30% of the size of the original device.

1. Introduction

Compact heat exchangers are important components of energy systems used in electronic cooling, heating, ventilation and air conditioning (HVAC), power generation, manufacturing and chemical processes, among several applications. In these thermal devices a common geometry is the plate-fin and tube configuration, a schematic of which is illustrated in Fig. 1 for the two possible tube arrangements: in-line and staggered. During its operation, the transfer of energy in a compact heat exchanger is between a liquid flowing inside the tubes and air flowing on the outside. Due to the thermo-physical properties of air, and frequent laminar nature of the flow, the resulting thermal resistance critically constraints the transfer of energy; the heat transfer area per unit volume is usually very large, and the attention by the scientific community has been devoted to seek for heat transfer enhancement in these systems.

To understand the process of convective heat transfer in these engineering equipment both numerical simulations and experimental studies have been carried out and reported in the open
literature; e.g., [1]–[3], with the idea of elucidating new ways to augment the transfer of energy [4]. Special interest has been placed in heat transfer enhancement through built-in obstacles [5], turbulators and winglets [6, 7], that enable generation of vortices to improve mixing in the fluid [8]. Few investigations have also looked at optimizing their location [9, 10].

In this study, we explore the problem from a different angle; that of increasing the efficacy of the device to design more compact devices, by identifying the regions in a complete device where most of the heat transfer takes place, and eliminating those that do not contribute significantly. In this way, a more compact heat exchanger can be designed and used to transfer the same amount of energy at the expense of less pumping power, hence larger energy savings. To facilitate the analysis, most studies of multiple-row compact heat exchangers have used a section of the computational domain imposing periodic conditions at the leading- and trailing-edge planes. In our case, we solve a more comprehensive section that includes the inlet and outlet sections of the heat exchanger along with the inner tubes. This approach is similar to those of Jang et al. [2] and Xie et al. [11], but the main difference is that we will use to heat transfer rate instead of the common Nusselt number as the basis for the calculations. This approach provides higher accuracy at the expense of less generality since the heat transfer coefficient is not needed [12, 13].

To this end, we perform three-dimensional numerical simulations on the external side of two plain-fin and tube configurations. The governing equations in Cartesian coordinates are first formulated and then solved on a representative computational domain for specific sets of flow and temperature values using a finite element method to compute the velocity, pressure and temperature fields. From these, heat rates and pressure drops are used to compare both types of tube-arrangement are compared to a baseline device to find the length needed to exchange a specific fraction of the amount of energy that would be transferred by the baseline model.

2. Problem description and governing equations

The typical in-line and staggered configurations of the plain-fin and tube compact heat exchanger considered here as the baseline for the analysis, which are common in HVAC applications, are shown schematically in Fig. 1. Regardless of the configuration, the heat exchanger has a nominal length of $L = 1016$ mm (101.6 cm), a height $H = 445$ mm and a width $W = 800$ mm, with an outside tube diameter $D = 31.8$ mm and a fin spacing $\delta = 4.4$ mm. Other important dimensions include the longitudinal fin pitch $P_l = 63.6$ mm, and the center-tube distances to the leading and trailing edges $L_L = 68.2$ mm and $L_T = 68.2$ mm. The value of the transverse tube pitch depends on the tube arrangement; $P_l = 63.6$ mm for in-line- and $P_l = 31.8$ mm for staggered-tubes. From the dimensions, the number of tube rows in the baseline heat exchanger is $N = 15$. Air is the working fluid flowing within the laminar regime on the external side, and water at high velocity (resulting in a constant tube-surface temperature), is assumed to flow inside the tubes.

Figure 2 illustrates a two-dimensional view of the sections of the corresponding heat exchanger geometry considered for the numerical simulations, where the focus is on the external side. Each shaded section represents the minimum domain capable of resembling the complete system. From
the figure it can be noticed that although we use the natural symmetry of the problem along the height and width (y and z-directions) dimensions. A major difference from other investigations is the use of the complete channel length (x-direction) for the analysis, instead of the typical periodic conditions. The air flow is in the x-direction.

![Diagram of heat exchanger computational domain](image)

**Figure 2.** Minimum heat exchanger computational domain.

The mathematical model comprises the continuity, momentum and energy equations for an incompressible, Newtonian fluid with constant properties, in the laminar regime, under steady-state conditions, and without body forces and viscous dissipation, can be written in vector form as

\[
\nabla \cdot \mathbf{u} = 0, \\
(\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u}, \\
\rho c (\mathbf{u} \cdot \nabla) T = k \nabla^2 T,
\]

where \( \mathbf{u} \) is the three-dimensional Cartesian velocity vector with components \( u, v \) and \( w \), in the directions \( x, y \) and \( z \), respectively; \( p \) is the pressure, \( \rho \) the density, \( \nu \) is the kinematic viscosity, \( c \) the specific heat, \( k \) its thermal conductivity and \( T \) is the air temperature.

Based on the minimum sections of Fig. 2, the boundary conditions for the flow of air over the tube bank and within the fin channels are the common no-slip and impermeable conditions at the tube and fin surfaces; uniform velocity \( u_{in} \) is invoked at the inlet cross section, and zero pressure with no viscous stresses are set at the outlet. As will be shown in the next section, to enforce these two boundary conditions with high accuracy, we follow Jang et al. [2] and Romero et al. [4], and use a computational domain that extends farther both upstream and downstream of the shaded heat exchanger sections. For the thermal boundary conditions the fin and tube surfaces \( T_s \) are assumed to be isothermal, the fluid entering the channel passage has a uniform temperature \( T_{in} \), and all temperature gradients are set to zero at the outlet. For both, velocity and temperature, symmetry invoked at the planes \( y = 0 \) and \( y = P_t/2 \) for in-line \( (y = P_t \) for the staggered case), and at the mid-plane \( z = 0 \). Considering a conjugate version of this problem
could slightly improve the calculations; however, it has been shown that the difference between ideal and real fins is usually small [14], and most calculations reported in the literature use this approach without significantly degrading the accuracy of the results.

3. Numerical methodology
The governing equations were discretized on the computational domain and solved by the finite element method. Although several other efficient numerical techniques based on finite difference approximations exist in the literature; e.g., [15], and have been successfully applied to a number of fluid flow and heat transfer problems [16]–[18] on systems with irregular geometries, the finite element method is advantageous in dealing with complex geometries, allowing easy specification of the boundary conditions on curved surfaces. The steady-state heat exchanger problem is solved with the general-purpose finite-element software COMSOL Multiphysics (http://www.comsol.com). The geometry is constructed using a three-dimensional mesh with quadrilateral elements along the boundaries and hexahedral elements inside the domain, where the degrees of freedom for temperature, velocity and pressure are all assigned at the nodes. To ensure accurate results while maintaining a manageable CPU time, more dense meshing is using near all the walls. A schematic of the typical discretization domain for both configurations is show in Fig. 3, where, for each section including a tube, it was characteristic to use 127 elements evenly distributed around the tube perimeter, being these mapped outward in a radial fashion to the outer edges of that section. These elements are then linked to a regular mapped mesh. The total number of elements for the in-line and staggered configurations was, respectively, 126656 and 126720. This difference was associated to the slightly larger \( L_{x2} \) that resulted for the staggered geometry. The velocity and temperature fields were determined by computing the system of algebraic equations simultaneously via the generalized minimum residual (GMRES) iterative solver, for which the relative tolerance was set to \( 10^{-6} \).

Several grids were tested for different values of the inlet velocity (i.e., Reynolds numbers), to ensure grid independence in the numerical results. As an example, Figure 4 illustrates a typical set of convergence tests for the pressure \( p \), and the streamwise velocity \( u \), at a fixed point in the domain —i.e., \((476.2, 15.9, 0) \text{ mm}—\) in both the in-line and staggered configurations, for an inlet velocity \( u_{in} = 0.23 \text{ m/s} \) (\( Re = 480 \) based on the tube diameter). From the figure, the expected error decay in both velocity and pressure, with respect to results from the finest mesh, is clearly seen with velocity values achieving convergence before those of the pressure. The figure also shows that regardless of the tube arrangement, a grid with less than 106000 elements is sufficient to achieve an accuracy within 2% of the results obtained with a grid containing 1.35 times as many elements. As previously mentioned, we have used a mesh containing close to 127000 elements, which ensures an excellent accuracy of less than 0.1% error.
Figure 4. Grid independence tests for $Re = 480$. In-line: -o- $u$; -<-> $p$. Staggered: -x- $u$; +-, $p$.

4. Parametric analysis and discussion
The numerical analysis is carried out for two parameters, named, the Reynolds number $Re = u_{in}D/\nu$, and the temperature difference between the fluid at the entrance and the heat exchanger surfaces $\Delta T = T_s - T_{in}$. The corresponding ranges are: $Re \in [120, 960]$ with four specific values, i.e., $Re = 120, 240, 480$ and $960$, and $\Delta T \in [5, 20]$ K in intervals of 5 K. Quantitatively the results shown in the next sections are based on average values of the streamwise fluid velocity, pressure and temperature, defined [19] as

$$
\bar{p} = \frac{1}{A} \int_A p \, dA; \quad \bar{\nu} = \frac{1}{A} \int_A (u \cdot \mathbf{n}) \, dA; \quad \bar{T} = \frac{\int_A (uT \cdot \mathbf{n}) \, dA}{\int_A (u \cdot \mathbf{n}) \, dA}, \quad (4)
$$

which can be computed once the values of velocity, pressure and temperature are obtained from solutions of the governing equations (1)–(3). In Eq. (4), $\mathbf{n}$ is the unit vector associated to the surface of interest $dA$; i.e., the cross section $dy \, dz$, at any point in the streamwise direction $x$. From the average values of velocity and temperature an energy balance provides the heat rate $Q$ between two selected sections as

$$
Q = \rho \bar{\nu} A c (\bar{T}(x) - \bar{T}(x + \Delta L)). \quad (5)
$$

The use of the heat rate, instead of the averaged $Nu$ number, is an innovative approach that provides higher accuracy since the heat transfer coefficient along with a characteristic temperature difference is not needed [12, 13].

In Eq. (5), $\Delta L$ is a specific section of the heat exchanger that may be selected by the user; i.e., the thermal engineer. The heat rate between the inlet and outlet sections of the heat exchanger is $Q_L = \rho \bar{\nu} A c(T_{in} - T_{out})$. As previously indicated, the baseline heat exchanger has $N = 15$ tubes in the streamwise direction.

4.1. In-line tube arrangement
For the baseline heat exchanger, typical velocity and temperature fields are shown in Fig. 5, as streamlines in Fig. 5(a), and isotherms in Fig. 5(b), at the mid-plane $z = 0$, for $Re = 960$.
and $\Delta T = 20$ K. Though not included here, results for other values of the parameters have shown similar behavior. From Fig. 5(a), the evolution of the flow patterns along the heat exchanger length can be clearly seen as the flow is uniform at the inlet and develops, due to the fin and tube interactions with the fluid, into periodic patterns in which the streamlines are closer to each other in regions where the tubes are present (indicating larger local velocities) and recirculation regions develop behind the tubes. At the outlet, this periodicity in the behavior is finally broken. On the other hand, influence of the hydrodynamics upon the transfer of energy is shown in Fig. 5(b), where it is clear that most of it is achieved in the first three tube sections. As before, the temperature at the entrance is uniform and the temperature field develops as the fluid travels along the heat exchanger, but not in a periodic pattern, with larger temperature changes achieved upstream and over the tubes. After approximately three sections the fluid temperature reaches the wall temperature of the fins and tubes.

![Streamlines](image1)

(a) Streamlines.

![Isotherms](image2)

(b) Isotherms.

**Figure 5.** Streamlines and Isotherms at the midplane for in-line configuration. $Re = 960$ and $\Delta T = 20$ K.

Figure 6 illustrates the pressure distribution along the channel streamwise direction [Fig. 6(a)], and the fraction of heat rate $Q$ that is transferred at different sections of the baseline heat exchanger [Fig. 6(b)], for $Re = \{120, 240, 480, 960\}$ and $\Delta T = 20$ K (as a representative example). From the figure it can be seen that as expected, the pressure decreases linearly along the channel, but at the same time, the pressure drop increases with increasing $Re$ number; i.e., inlet velocity of the air flowing into the channel of the heat exchanger. Also shown in the figure is the fact that, regardless of the value of $Re$, there is a rapid decrease in the amount of energy transferred within the system as the air temperature quickly reaches the wall temperature. It can be noticed, however, that the actual streamwise location (length) at which the heat transfer rate can be considered as negligible depends on the $Re$ number; the $x$-location increases with increasing $Re$ since the residence time of air in contact with the heat exchanger walls decreases. It is important to note that for other values of $\Delta T$, the $Q-x$ curves are qualitatively similar, the only difference being the actual $x$-location at which the heat transfer is no longer appreciable. For $Re = 960$, which is the critical case, this location is within 30% of the total length $L$.

4.2. Staggered tube arrangement

The corresponding numerical results for the staggered arrangement are shown in Fig. 7. Again, the streamlines and isotherms at the mid plane are shown, respectively, in Figs. 7(a) and 7(b), for $Re = 960$ and $\Delta T = 20$ K. From the figures it can be observed that similarly with the in-line configuration, there are periodic patterns in the flow which are broken at the leading and trailing edges of the heat exchanger and recirculation bubbles behind the tubes. This time, however, the patterns alternate due to the tube arrangement, hence improving mixing in the fluid and increasing the transfer of energy. The isotherms shown in Fig. 7(b) clearly show the influence of the flow pattern as they develop in an alternate fashion. As before, only three tube sections are necessary for the air temperature to reach the temperature at the walls, hence decreasing the amount of heat transfer convected downstream of these.
Figure 6. Parametric analysis for in-line configuration. -○- Re=120; -×- Re=240; -□- Re=480; -+- Re=960.

Figure 8 illustrates the pressure and heat rate distributions along the streamwise direction of the staggered configuration. As in the previous section, both pressure and heat rate show similar trends to their in-line counterparts. For pressure, the main difference is the slightly larger values of the pressure drop for the staggered case. For the case of the heat rate, larger values are achieved at streamwise locations closer to the leading edge, but becoming negligible at a location in $x$ very close to 30% of the length $L$.

Figure 7. Streamlines and Isotherms at the mid-plane for staggered configuration. $Re = 960$ and $\Delta T = 20$ K.

Since the main objective is to design more compact heat exchangers, a comparison between the in-line and staggered configurations, for the fraction of the total heat transfer that can be achieved by the baseline device, and the corresponding length (given in terms of the number of tubes in the section), is shown in Tables 1 and 2 for the set parameter values $Re = \{120, 240, 480, 960\}$ and $\Delta T = \{5, 10, 15, 20\}$ K. From the tables it can be seen that in general, the number of tubes $N_{tubes}$ necessary to achieve a specific fraction of $Q_{total}$, i.e., $\sum Q$, depends on the $Re$ number and the corresponding tube arrangement. Thus, for $Re$ numbers in the range 120 to 240, regardless of $\Delta T$, and for fractions of the total heat rate from 90 to 99%, the number of tubes is constant and equal to 1. However, as the $Re$ number increases beyond 240, regardless of the arrangement, the number of tubes also increases in direct relation to the higher fraction of the total heat rate. Finally, at large $Re$ numbers, the staggered configuration requires a smaller amount of tubes to achieve at least 97% of the total possible transfer of energy...
as compared to the in-line arrangement; this corresponds to less than 30% of the length of the baseline device.

Table 1. Number of tubes associated to percentage of $\sum Q/Q_{\text{total}}$ for in-line configuration.

| $Re$ | $\Delta T$ (K) | $N_{\text{tubes}}$ |
|------|----------------|-------------------|
| 120  | 5,10,15,20     | 1 1 1 1           |
| 240  | 5,10,15,20     | 1 1 1 1           |
| 480  | 5,10,15,20     | 1 1 1 2           |
| 960  | 5,10,15,20     | 1 2 3 4           |
|      | $\sum Q/Q_{\text{total}}$ (%) $\rightarrow$ | 90 95 97 99 |

Table 2. Number of tubes associated to percentage of $\sum Q/Q_{\text{total}}$ for staggered configuration.

| $Re$ | $\Delta T$ (K) | $N_{\text{tubes}}$ |
|------|----------------|-------------------|
| 120  | 5,10,15,20     | 1 1 1 1           |
| 240  | 5,10,15,20     | 1 1 1 1           |
| 480  | 5,10,15,20     | 1 1 1 2           |
| 960  | 5,10,15,20     | 1 2 2 3           |
|      | $\sum Q/Q_{\text{total}}$ (%) $\rightarrow$ | 90 95 97 99 |

5. Concluding remarks

Parametric analyses, via numerical simulations, provide an advantageous approach for obtaining accurate solutions of complex engineering systems, which otherwise would be extremely difficult.
to obtain experimentally under reasonable allocation of time and resources. In this investigation we have carried out fluid flow and heat transfer simulations on the external side of a plain-fin and tube heat exchanger to analyze the interaction between fluid and the geometry towards the design of more compact devices. Finite-element-based computations were performed for several Reynolds numbers in the laminar regime. For the two configurations analyzed, i.e., in-line- and staggered-tube arrangements, results from this investigation indicate that the geometry plays a major role in the amount of heat being exchanged and that, for a given device, the length needed to exchange 99% of the corresponding amount of energy that may be transferred by the baseline model, is confined to less than 30% of the size of the original device.

Acknowledgments
A. Motamedi is the recipient of a CEaS fellowship for which we are grateful. The authors wish to acknowledge the contribution of L. Ramirez-Gonzalez in the early stages of the development of this work. This research has been partially supported by NSF IIP-0724505, IIP-0844891 and HRD-0932421 grants.

6. References
[1] Bastani A, Fiebig M and Mitra N 1992 Design and Operation of Heat Exchangers ed Roetzel W, Heggs P and Butterworth D (Berlin: Springer-Verlag) pp 154–163
[2] Jang J Y, Wu M C and Chang W 1996 Int. J. Heat Mass Transfer 39 3057–3066
[3] Sunden B 2010 Int. J. Numer. Methods Heat Fluid Flow 20 551–569
[4] Romero-Mendez R, Sen M, Yang K and McClain R 2000 Int. J. Heat Mass Transfer 43 39–51
[5] Leu J, Wu Y and Jang J 2004 Int. J. Heat Mass Transfer 47 4327–4338
[6] Vasudevan R, Eswaran V and Biswas G 2000 Numer. Heat Transfer, Part A 51 533–555
[7] Joardar A and Jacobi A 2007 ASME J. Heat Transfer 129 1156–1167
[8] Valencia A, Fiebig M and Mitra N 1996 ASME J. Heat Transfer 118 200–211
[9] Chu P, He Y, Lei Y, Tian L and Li R 2009 Appl. Thermal Engineering 29 859–876
[10] Lemouedda A, Breuer M, Franz E, Botsch T and Delgado A 2010 Int. J. Heat Mass Transfer 53 3586–3599
[11] Xie G, Wang Q and Sunden B 2009 Int. J. Heat Mass Transfer 29 1–16
[12] Pacheco-Vega A, Díaz G, Sen M, Yang K and McClain R 2001 ASME J. Heat Transfer 123 348–354
[13] Pacheco-Vega A, Sen M and Yang K 2003 Int. J. Heat Mass Transfer 46 1029–1040
[14] R R M, R A and M S 2001 Interiencia 26 321–326
[15] Pacheco J 2001 Int. J. Numer. Methods Fluids 35 71–91
[16] Pacheco J and Pacheco-Vega A 2003 J. Fluids Engrg. 125 365–374
[17] Pacheco J, Pacheco-Vega A, Rodić T and Peck R 2005 Numer. Heat Transfer, Part B 48 1–24
[18] Pacheco-Vega A, Pacheco J and Rodić T 2007 ASME J. Heat Transfer 129 1506–1516
[19] Lee K S, Kim W S and Si J M 2001 Int. J. Heat Mass Transfer 44 3223–3231