Thermal Characterization of Open Lattice Structures used as Heat Exchanger Surfaces

J. Gullbrand*, K. Balantrapu† and R.A. Wirtz‡
Mechanical Engineering Department, University of Nevada, Reno, Reno, NV 89557

Numerical simulation is performed to evaluate flow field and thermal characteristics in heat exchange matrices. The heat exchanger geometry evaluated consists of an isotropic open lattice structure containing group-interconnected, millimeter-scale, thermally conductive ligaments. The Reynolds number range of interest is Re = 3 - 1000. The flow fields investigated in this paper are both laminar and turbulent. The laminar flow fields are solved using direct numerical simulation (DNS), while turbulent flow fields are solved by large-eddy simulation (LES). The mean velocity profiles and velocity fluctuations are compared to each other and to experimental data for a packed bed of spheres. Friction factor and Colburn j-factor are calculated from the simulated flow fields and averaged in time. They are compared to the algebraic correlations reported in the literature for a packed bed of spheres and to experimental data for a screen laminate.

Nomenclature

c = specific heat coefficient of the fluid.
d = diameter of the thermally conductive ligaments.
Dh = hydraulic diameter.
f = friction factor.
G = spatial filter function.
j = Colburn j-factor.
p = pitch between the thermally conductive ligaments in each spatial direction.
P = pressure.
Pr = Prandtl number.
q'' = heat flux.
Re = Reynolds number based upon the average pore velocity and hydraulic diameter.
s+ = normalized distance from the wall s+/uτ, s is the distance from the wall and uτ is the friction velocity.
Sij = strain rate tensor.
St = Stanton number based upon the temperature difference between the fluid inlet to solid interface.
T = temperature.
Tf,i, Tf,o = temperature of the fluid entering and leaving the heat exchanger respectively.
Tin = average inlet temperature to the unit cell.
ui = flow velocity in the spatial i-direction.
vrms = root-mean-square velocity fluctuations including contribution from all three spatial fluctuations.
V = average pore velocity.
Vi = superficial velocity.
β = heat transfer surface area-to-volume ratio.

* Research Assistant Professor, email: jgullbra@unr.edu, AIAA member.
† Graduate Student, AIAA member.
‡ Professor, email: rawirtz@unr.edu, AIAA member.

Copyright © 2005 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.
I. Introduction

The most efficient method to increase the performance of a heat exchanger is to increase the surface area-to-volume ratio $\beta$. Small-particle packed beds and foamed metals have a large surface area-to-volume ratio, and therefore seem to be good candidates for compact and small-scale heat exchangers. However, the effective thermal conductivity is relatively small for these structures. The effective thermal conductivity is in the range of 10 - 15% of the particle thermal conductivity for a packed bed of spheres, while it is only 2 - 6% of the base metal value for a commercially available aluminum foam. Therefore, most of the gain in performance obtained by the increased surface area-to-volume ratio is lost by the lowering of the effective thermal conductivity.

Effective heat exchangers can be created if the effective thermal conductivity can be increased in the porous media structures with a large area-to-volume ratio. Increasing the effective thermal conductivity can be done by using uninterrupted filaments in the direction of preferred heat transfer. This will result in a directional heat exchanger. Examples of structures with these characteristics are stacked two-dimensional woven screens, and three-dimensional woven mesh structures. In both of these cases, the effective thermal conductivity in one particular direction is approaching 78% of the base material value. The geometrical configurations of the structures can be varied, and they can provide a wide range of porosity and specific surface area.

The design of these structures allows for single-fluid, compact heat exchangers that possess structural functionality, so they can be deployed in a range of multifunctional applications. The high directional thermal conductivity allows the heat exchanger to be used when highly localized spot cooling is required. The structure can also be incorporated into the design of a flow through module as well as to replace the fin-array in a plate-fin heat exchanger. These heat exchanger structures can also be easily conformal to complex surfaces. The goal of this analysis is to obtain structures that have high thermal performance and a reasonable pressure drop.

In the investigation presented here, an open lattice structure is used as the heat exchanger surface. The lattice structure is a modification of the two-dimensional screen laminates that was investigated previously by Ref. 11, and the three-dimensional mesh studied by Ref. 15. The open lattice structure is a three dimensional structure that consists of group interconnected thermally conductive ligaments. The shape of the conductive ligaments can be varied to fulfill different design criteria of the heat exchanger, and they can be different in each spatial direction. This creates a structure with huge flexibility in reaching the design criteria.

The Reynolds number range of interest for the open lattice heat exchangers varies from laminar to turbulent flow fields inside the porous media. The range of interest is between $Re = 3$ and 1000. The Reynolds number is calculated by $Re = \frac{\rho V D_h}{\mu}$, where $\rho$ is the density of the fluid, $V$ is the average pore velocity, $D_h = 4\varepsilon/\beta$ is the hydraulic diameter, and $\mu$ is the molecular viscosity. Flow regimes in porous media can be divided into four different regions according to Ref. 4, who investigated a packed bed of spheres: creeping flows (Darcy flows), $Re < 1$; laminar flow, $1 < Re < 150$; unsteady laminar flow, $150 < Re < 300$; and, turbulent flow $Re > 350$.

In order to capture any unsteadiness in the laminar flow fields, time resolved simulations are required. Therefore, in this study the low Reynolds number simulations are performed using direct numerical simulations (DNS). For simulation of the turbulent flow fields, large-eddy simulations (LES) are performed. In LES the important large, energy-carrying length-scales are resolved, while the small unresolved scales are modeled with subgrid-scale (SGS) models.

In the literature, studies have been performed to evaluate the flow and thermal characteristics of porous media flows both numerically and experimentally. One porous media, which is easy to define and is well documented, is the packed bed of spheres. A typical porosity of a packed bed of spheres is $\varepsilon = 0.4$, and therefore, this is the porosity that is used in our open lattice simulations. Experimental measurement of velocity characteristics for a packed bed of spheres are reported for laminar flows by Ref. 13, while data for turbulent flows are reported by Ref. 15. Most turbulent simulations that have been reported in the literature are based upon Reynolds Averaged Navier-Stokes (RANS)
The RANS simulations produce statistically averaged flow fields. TheLES results obtained in this study will also give time resolved information about the flow and thermal characteristics.

The characteristics of open lattice structures are not well documented, neither experimentally nor numerically. Therefore, this investigation is the first step in documenting the characteristics of the flow field in an open lattice structure. The geometry consists of interconnected circular cylinders of the same diameter (d) and pitch (p) such that p/d = 1.55 leading to a porosity of \( \varepsilon = 0.4 \) and \( \beta d = 2.73 \). The velocity profiles are compared to each other and to experimental data for a packed bed of spheres. The performance of the open lattice heat exchanger is determined by studying the friction factor and the Colburn j-factor. These two quantities are compared to correlations found in the literature valid for a packed bed of spheres and to the experimental data of the woven screen laminates by Ref. 11.

II. Numerical methods

A. Governing Equations

The governing equations for an incompressible and thermal flow field are the continuity equation, the Navier-Stokes (N-S) equations, and the energy equation. Written in Cartesian coordinates, the equations are:

\[
\frac{\partial u_i}{\partial x_i} = 0,
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j^2}, \text{ and } \frac{\partial T}{\partial t} + u_j \frac{\partial T}{\partial x_j} = \frac{k}{\rho c} \frac{\partial^2 T}{\partial x_j^2} \tag{1, 2, 3}
\]

In LES, the governing equations are filtered in space. The filtering procedure is applied to every flow field variable according to:

\[
\bar{u}_i = \int_{-\infty}^{\infty} G(x, x', \Delta) u_i(x', t) dx',
\]

where \( G \) is the filter function and \( \Delta \) the filter width. The filtering of the variables can be applied either implicitly or explicitly. The most commonly used filtering procedure is the implicit one, where the computational grid and the discretization operators are considered as the filtering of the governing equations. In the explicit procedure, an explicit expression of the filter function is used. In this investigation, the implicit filtering procedure is applied. The filtered governing equations can be written as

\[
\frac{\partial \bar{u}_i}{\partial x_i} = 0, \quad \frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j^2}, \text{ and } \frac{\partial T}{\partial t} + \bar{u}_j \frac{\partial T}{\partial x_j} = \frac{k}{\rho c} \frac{\partial^2 T}{\partial x_j^2} \tag{5, 6, 7}
\]

where the turbulent stresses are defined as \( \tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \) in the momentum equation and the energy transport tensor in the energy equation is \( \sigma_{ij} = \bar{u}_i \bar{T} - \bar{u}_j \bar{T} \). The filtered N-S equations and energy equations are not closed due to the non-linear terms that need to be modeled; i.e. \( \bar{u}_i \bar{u}_j \) in the momentum equation and \( \bar{u}_i \bar{T} \) in the energy equation.

B. Subgrid-Scale Models

The aim of subgrid-scale modeling is to accurately capture the influence from the small unresolved scales on the resolved ones by using the resolved flow field quantities. Several SGS models have been proposed in the literature, but the eddy viscosity model proposed by Ref. 12 is the most commonly used. The Smagorinsky model (SM) is

\[
\tau_{ij} = -2\nu_s \bar{S}_{ij} = -2(C_s \Delta)^3 \left| \bar{S} \right| \bar{S}_{ij} = -2(C_s \Delta)^3 (2\bar{S}_{ij}\bar{S}_{kl})^{1/2} \bar{S}_{ij},
\]

where \( \nu_s \) is the turbulent eddy viscosity, \( C_s \) is the Smagorinsky constant, and \( S_{ij} \) the strain rate tensor. The SM is purely a dissipative model that is easy to implement and has shown to perform reasonably well in a variety of flows. However, the model is not universal and the model coefficient \( (C_s) \) used needs to be changed for different flow fields. Ref. 5 proposed a dynamic procedure to recalculate the model coefficient during the simulation to depend upon local flow field characteristics. In the dynamic procedure, the model parameter, or actually the product \( (C_s \Delta) \), is recalculated during the entire simulation. The advantage of including the length-scale \( \Delta \) in the dynamic procedure is to avoid the ambiguity associated with determining the appropriate length-scale especially when using an unstructured computational grid. In the dynamic procedure, a test filter function is used that has a larger filter width than the filter function applied to the governing equations. In computations that applies unstructured computational grids, a volume average is usually applied as the test filter function. In the simulations presented in this paper, a
volume average of the flow variables is used, and also the model coefficient is volume averaged before it is applied. The least-square method suggested by Ref. 8 is used to determine one model parameter from the six independent equations. The minimum viscosity in the simulations is limited to zero and therefore, the smallest value the eddy viscosity can attain is \( -\nu \).

C. Computational Code

Fluent (version 6.1.22) is used in the current simulations. The flow solver is based upon finite volume collocated second-order spatial discretization. In the simulations presented here, second-order central differencing has been used for the convective terms. The SIMPLEC algorithm is used to enforce the coupling between the velocity and pressure field and to impose the divergence-free condition. A dual time stepping technique is employed to advance the simulation in time. The dual technique consists of an outer and an inner loop, where the outer loop is based upon a three-level second-order time advancement, while the inner iteration uses an algebraic solver with a Gauss-Seidel smoother. Ten inner iterations are used for each time step. The time advancement is continued until time averaged mean flow quantities are obtained that no longer change.

III. Computational Test Case

A. Heat Exchanger Implementation

An open lattice structure can be implemented in a parallel-plate channel with height 2H (Fig. 1). In this application, heat will be conducted from the exchanger plates (laying in the x-z plane), along y-filaments and then to the coolant flowing through the array. The filaments in the x- and z-directions act as fins, increasing the surface area of the structure. The open lattice structure acts as a porous wall with thickness t. The coolant approaches the lattice structure at the superficial velocity \( V_i \) and temperature \( T_{f,i} \). The superficial velocity is the same after the heat exchanger, but the temperature of the coolant has changed to \( T_{f,o} \). The superficial velocity is related to the average pore velocity \( V \) through the porosity, \( V = V_i/\varepsilon \).

B. The Open Lattice Structure

The open lattice structure consists of group-interconnected mm-scale thermally conductive ligaments. In this study, the conductive elements are mutually perpendicular circular cylinders as shown in Fig. 2. The filament pitch is \( p \), and the filament diameter is \( d \). The pitch is the same in all spatial directions and chosen to be \( p = 3.1 \) mm. The diameter are the same for all cylinders and chosen to \( d = 2.0 \) mm. However, the results presented in this paper scale with the geometry and are valid for all isotropic circular cylinder lattice structures with a porosity of \( \varepsilon = 0.4 \) and \( \beta d = 2.73 \).

C. The Flow Domain and Boundary Conditions

Since heat exchanger matrices consist of a large number of cells (or elements), and it is not feasible to resolve the flow field using the whole structure; rather one “unit cell” flow volume of the matrix, of dimension \((p_x, p_y, p_z)\), is used in the simulations. Studying flow field and thermal characteristics of the unit cell is expected to give a good representation of the performance of the heat exchanger. However, the study of the single element will not show any entrance or wall effects. With a packed bed of spheres, wall effects are shown to influence the flow characteristics.
for the first 1.5 diameters from the boundary walls, while entrance effects are registered up to the order of magnitude of five diameters.

The solution domain used in the numerical simulations is shown in Fig. 3. The outer dimensions of the domain have edge length \((p_x, p_y, p_z)\). Concave surfaces are fluid-solid interfaces having radius equal to the radius of the cylindrical filaments. Plain surfaces are openings that connect to adjacent unit cell fluid volumes. The simulations are performed by applying a specified mass flow through the domain in the \(x\)-direction. Periodic boundary conditions are applied in the \(x\)-direction to enforce global mass conservation; the same amount of mass flow entering through the inlet plane also leaves through the outlet. The periodic boundary condition is also used for the lateral plane surfaces that are not in the main flow direction. These periodic conditions enforce mass conservation, and do allow the domain to interact with surrounding fluid domains in the heat exchanger. If fluid is entering through one surface, the same amount must also leave the opposite surface.

Fluid can therefore pass in or out of the domain through the lateral planes that are not in the main flow direction. In the simulations, the fluid is water with \(Pr = 7.0\).

The thermal boundary conditions for the openings are periodic as well. In the simulations, the fluid-solid interfaces are assumed to be isothermal. Even though the temperature drop over the whole exchange matrix may be substantial, the solid phase is approximated to be isothermal due to the unit cell’s small length scale (mm-scale). This approximation can be justified by a simple scale analysis: Assume that the temperature difference between the wall and the approaching coolant is \(T_w - T_{ci} = 10^\circ C\) (Fig. 2), and the temperature drop across a 1 cm exchange matrix is \(1^\circ C\) (90% fin efficiency), then the temperature drop across a 1mm feature of the exchange matrix is 0.1 \(^\circ C\) or 1% of the driving temperature difference. All the simulations are started from a zero velocity field. Thereafter, the simulations are advanced in time. The low Reynolds number flows converge to a steady solution, while the high Reynolds number flows break down to unsteady flows.

D. Computational Grid and Convergence Criteria

The computational grid of the flow domain consists of boundary layer grids in the near wall region of the flow, and an unstructured grid in the computational zones far from the walls. The boundary layer grids are needed to resolve the flow physics in the near wall region. The grid resolution in the outer part of the flow does not need to be as fine as in the near wall region. The total number of node points needed in the simulations are different in each simulation. In all simulations, the flow fields are resolved in time, and the laminar flow fields converge to a steady flow. One computational grid is used to resolve the laminar flow field at a specific Reynolds number. Thereafter, the computational grid is refined, and the simulations are redone. The friction factor is calculated from the simulated results. If the friction factor is not changing more than 1% in the two computational grids, the simulated results are determined to be grid converged. A criterion of 10% is applied in the thermal flow field simulations, where the Colburn j-factor is used as the variable of convergence. Boundary layer grids are used on all the wall/fluid boundaries to resolve the near wall flow region of the unsteady flow fields. The boundary layer grids are employed to ensure that the first computational grid point is located at a distance of \(s^+ \leq 1\) from the wall. This is the resolution that is required to resolve the important near wall region. The simulated data is averaged over time to determine the mean velocity profiles, velocity fluctuations, friction factor, and Colburn j-factor.

IV. Results

A. Velocity and Temperature Contours

The velocity contours for Re = 100 are shown in Fig. 4 at two different cross-sections of the flow domain. The cross-sections are centered in the flow domain. The flow is in the positive x-direction. The velocity contours are normalized with the average pore velocity. The flow is steady and laminar for Re = 100. The figures clearly show a jet-like flow field through the domain. In the right figure, the inlet is on the left side of the domain and the outlet at the right side. The jet expands as the cross-sectional flow area increases, and then contracts as the area decreases as
the flow reaches the outlet of the domain. The left figure shows a cross-section of the jet when the stream-wise direction is into the plane. This figure shows that the jet does not have a circular shape, but more of a rectangular one, where the corners of the rectangle are shifted 45° off the coordinate axis. The flow areas outside of the jet experiences a negative stream-wise velocity.

Temperature contours are shown in Fig. 5 at the same two locations as the velocity contours in the previous figure. The temperature is normalized according to \((T-T_{in})/(T_w-T_{in})\), where the \(T_{in}\) denotes the average inlet temperature, and \(T_w\) the isothermal temperature of the walls. The jet like character is clearly visible also for the temperature field. The areas outside of the jet experience a higher temperature than the jet itself, since the temperature of the walls are higher than the average inflow temperature. However, the four temperature regions outside of the jet (the four arms of the contour) indicates that recirculation zones are present. The recirculation zones are circulating low temperature fluid that comes from the jet into a region of higher temperature, where it is heated up.

The unsteady simulations (\(Re = 300\) and \(Re = 1000\)) show the same quantitative flow and thermal behavior in mean flow quantities as the laminar steady simulations do. The instantaneous picture of the flow however is looking very different (Fig. 6). The unsteady flow field clearly shows a range of length-scales present in the unsteady jet.

B. Mean Velocity Profiles and Velocity Fluctuations

The mean velocity profiles are plotted as a function of the distance from the center-point of the flow domain for different Reynolds number simulations. The velocity profiles are obtained along line AA shown in Fig. 4. The center of the jet is the reference location (zero location) in Figs. 7 and 8 and the velocity profiles are plotted in the \(y\)-direction (transverse direction). The results are all normalized with the average pore velocity (\(V\)).

In Fig. 7, the stream-wise velocity profiles of the laminar steady simulations are shown. The simulations converge to a steady solution for all the Reynolds numbers reported in the figure, from \(Re = 3\) up to \(Re = 100\). This is consistent with the findings of Ref. 4, who saw no instabilities below \(Re = 150\). The peak value of the mean velocity profiles increase as the Reynolds number of the flow increases and the jet like behavior becomes more pronounced. The same behavior was also reported by Ref. 13 for a packed bed of spheres, but the normalized peak values are higher in the open lattice structure than in a packed bed of spheres.

The mean velocity profiles of the stream-wise velocity for two unsteady flows are shown in Fig. 8. The results from the \(Re = 300\) flow falls into the unsteady laminar regime proposed by Ref. 4. This seems to be a likely classification of the present flow field. The flow field is clearly unsteady and large unsteady structures are observed in the flow. A RANS simulation was performed (Spalart-Allmaras one-equation model used) of the flow field to determine the ratio between the turbulent viscosity and the molecular viscosity. This viscosity ratio can then be used to determine if the flow field is turbulent or not. The maximum ratio came out to be 5, which indicates that the flow field is not turbulent. On the other hand, the \(Re = 1000\) flow is most likely a turbulent flow field, but not fully developed. The same RANS simulation was performed for \(Re = 1000\) and the maximum ratio between the turbulent
and molecular viscosities are 25. This ratio is higher than for Re = 300 but it indicates that the flow field is not fully turbulent. However, the dynamic Smagorinsky model is used only in the LES calculations of Re = 1000, and not in the calculations of the unsteady flow of Re = 300. The mean velocity profile is broader for the Re = 1000 than in the Re = 300 simulation. The peak value of the mean velocity profiles shows a reversed dependence upon the Reynolds number when compared to the steady laminar profiles.

The lower curves in Fig. 8 show the root-mean-square (rms) value of the velocity fluctuations. The fluctuations obtained in the Re = 1000 simulations have a maximum normalized value of 0.93 (normalized with the average pore velocity), which results in fluctuations of 26% of the maximum mean velocity. These results are consistent with the findings of Ref. 17, who observed maximum rms fluctuations of about 20% of the mean velocity profile in a packed bed of spheres for Re = 6300. Two peaks are observed in the velocity fluctuations, where the shear stress is largest.

B. Friction factor and Colburn j-factor

Fig. 9 shows the friction factor as a function of Reynolds number. The friction factor is calculated according to $f = (2\Delta p D_h)/(\rho p V^2)$, where $\Delta p$ is the pressure drop, $D_h$ is the hydraulic diameter, and $p$ is the pitch. The friction factor of the open lattice structure is compared to two references. The first reference is the Ergun correlation for a packed bed of spheres$^1$, and the second reference is experimental data for woven stacked screen laminates from Ref. 11. The porosity of the screen laminates are $\varepsilon = 0.69$ and the experimental data is obtained for a number of ten stacked screens. The experimental data is expected to include entrance and wall effects, while the simulated results do not. The experimental data from Ref. 11 has been corrected to $\varepsilon = 0.4$ in Figs. 9 and 10.

The friction factor calculated from the simulated results for the open lattice structure show a lower friction factor than for both the packed bed of spheres and the screen laminates. The friction factor decreases as the Reynolds number increases for the low Reynolds number laminar flow fields. As the flow fields become unsteady and/or turbulent with an increasing Reynolds number, the friction factor levels out and becomes constant. The experimental data includes entrance effects, which increases the friction factor.

The Colburn j-factor as a function of Reynolds number is shown in Fig. 10. In the numerical simulations, the Colburn j-factor is calculated according to $j = S\theta \Pr^{2/3}$, where $S\theta$ is the Stanton number based upon the temperature difference between the fluid/solid interface and the average inlet flow temperature. The calculated Colburn j-factor for the open lattice structure is compared to two different reference cases. The first reference is an algebraic correlation function for a packed bed of spheres, Park et al. 2002. The screen laminates and the open lattice structure are compared in Fig. 10.
spheres proposed by Ref. 11. The second reference is experimental data of the screen laminates by Ref. 11. The experimental data was obtained for a constant heat flux for a stack of ten screens.

The Colburn j-factor results for the open lattice structure are between the results from the packed bed of spheres and the screen laminates. The simulated results are closer to the data for the screen laminates for the low Reynolds number flows (laminar flows, Re = 3 - 100). A jump in the simulated values of the Colburn j-factor is observed as the flow field becomes unsteady (Re = 300). The data points denoted with a solid square for the unsteady flow fields in Fig. 10 are the result of solving the filtered governing equations (Equations 5, 6, and 7) simultaneously and averaging the Colburn j-factor over time. In the simulations, the same time step and number of inner iterations are used for solving the N-S equations and the energy equation. The data denoted by the open squares represent results from simulations where the energy equation is solved for a frozen flow field. The data calculated for several different flow fields are then averaged. These simulations result in a lower value of the Colburn j-factor. The differences observed in the data between the two approaches are due to the time derivative of the temperature field ($\frac{\partial T}{\partial t}$). The reason for performing the simulations of solving the energy equation separately was to determine the influence from the time term in the energy equation and to verify that the jump of the j-factor observed for the unsteady flow fields was not due to the handling of the time term. The values of the Colburn j-factor from solving the equations simultaneously might be slightly reduced as the time step is decreased and the number of inner iterations is increased. However, the value is expected to be somewhere in the range between the solid and open square value. This needs to be determined in future simulations. The unsteady flow result is a preferred behavior of the open lattice structure used as a heat exchanger surface, since the friction factor is relatively low and the Colburn j-factor is relatively high.

V. Conclusions

Numerical simulations are performed of the flow and thermal characteristics in an isotropic open lattice structure ($\varepsilon = 0.4$ and $\beta d = 2.73$) to determine the performance of the structure as a heat exchange surface. The simulations deal with a specific diameter and pitch. However, the results scale with the geometry and are valid for all isotropic circular cylinder lattice structures with a porosity of $\varepsilon = 0.4$ and $\beta d = 2.73$. The open lattice structure consists of inter-connected thermally conductive ligaments. The ligaments are circular cylinders and the all have the same diameter. The simulations are time resolved to capture any unsteadiness in the flow fields. The low Reynolds number flows were simulated using direct numerical simulations, while the unsteady flows were simulated using large-eddy simulations.

The mean velocity profiles and velocity fluctuations were studied in the pores of the heat exchanger. The flow and thermal fields were simulated for six different Reynolds numbers, Re = 3, 10, 30, 100, 300, and 1000. The four lowest Reynolds numbers resulted in steady laminar flow fields. The findings are consistent with the results reported by Ref. 4 for a packed bed of spheres. The flow fields of the two highest Reynolds numbers resulted in unsteady flows. Simulations using Reynolds-Averaged Navier-Stokes equations were used to determine if the flow fields where turbulent or unsteady laminar. Re = 300 was found to most likely be an unsteady laminar flow, while Re = 1000 show tendencies to be a turbulent flow.

The flow field through the lattice structure showed a jet-like behavior. The steady laminar flow fields (the four lowest Reynolds numbers) showed a peak value of the mean velocity profile that increased with an increasing Reynolds number. The two unsteady flows investigated (the two largest Reynolds numbers), showed reversed dependence upon the Reynolds number. The peak value decreased as the Reynolds number increased and the velocity profiles broadened. However, the velocity fluctuations increased with increasing Reynolds number. Velocity fluctuations of 26 % of the maximum mean velocity was observed for Re = 1000.

The performance of the open lattice structure as a heat exchanger surface showed promising results. The preference for a heat exchanger is to have low friction factor (low friction losses) and high Colburn j-factor (high
heat transfer between the fluid and the solid structure). The fluid and thermal fields in the open lattice structure result in a lower friction factor and in a higher Colburn j-factor than for stacked woven screen laminates.

The ongoing research efforts for the open lattice structures used as heat exchanger surfaces are continued with investigating anisotropy effects and change of shape of the conductive ligaments. The anisotropy effects will be studied by increasing the diameter of the circular cylinder in the preferred heat transfer direction, and keeping the ligaments the same in the two other spatial directions. The advantage with this anisotropic structure is that the effective thermal conductivity is increased in the preferred heat transfer direction. The pressure drop and therefore the friction factor can most probably be decreased even further if the conductive ligaments are changed to elliptical shaped cylinders. The ellipses will be streamlined in the flow direction. These changes to the open lattice structure shows great promise to construct even more efficient compact heat exchangers.

Acknowledgments

The Missile Defense Agency through the Air Force Office of Scientific Research, USAF, sponsors this work under contract number F49620-03-1-342. The views and conclusions contained herein are those of the authors and should not be interpreted as necessarily representing the official policies or endorsements, either expressed or implied, of the Missile Defense Agency, the Air Force Office of Scientific Research, or the U.S. Government.

References