Proceedings of Fifth International Conference on **Enhanced, Compact and Ultra-Compact Heat Exchangers: Science, Engineering and Technology**, Eds. R.K. Shah, M. Ishizuka, T.M. Rudy, and V.V. Wadekar, Engineering Conferences International, Hoboken, NJ, USA, September 2005.

CHE2005 - 14

DESIGN CONSIDERATIONS FOR COMPACT CERAMIC OFFSET STRIP -FIN HIGH TEMPERATURE HEAT EXCHANGERS

Sundaresan Subramanian[1,2], Valery Ponyavin[1,2], Clayton Ray DeLosier[1,2], Yitung Chen[1,2]
Anthony E. Hechanova[3]

[1]Nevada Center for Advanced Computational Methods, University of Nevada, Las Vegas
[2]Department of Mechanical Engineering, University of Nevada, Las Vegas
[3] Harry Reid Center for Environmental Studies, University of Nevada, Las Vegas
E-mail: uuchen@nscee.edu

ABSTRACT

This paper deals with the development of a three-dimensional numerical model to predict the overall performance of an advanced high temperature heat exchanger (HTHX) design, up to 1000°C, for the production of hydrogen by the sulfur iodine thermo-chemical cycle used in advanced nuclear reactor concepts. The design is an offset strip-fin, hybrid plate compact heat exchanger made from a liquid silicon impregnated carbon composite material. The two working fluids are helium gas and liquid salt (FLINAK). The offset strip-fin is chosen as a method of heat transfer enhancement because of its ability to induce periodic boundary layer restart mechanism between the fins that has a direct effect on heat transfer enhancement. The effects of the fin geometry on the flow field and heat transfer are studied in three-dimensions using Computational Fluid Dynamics (CFD) techniques, and the results are then compared with the results from the analytical calculations. The pre-processor GAMBIT is used to create a computational mesh, and the CFD software package FLUENT that is based on the finite volume method is used to produce the numerical results. Fin dimensions need to be chosen that optimize heat transfer and minimize pressure drop. Comparisons of the overall performance between the rectangular and curved fin geometry were performed using computational fluid dynamics techniques. The model developed in this paper will be used to investigate the heat exchanger design parameters in order to find an optimal design. Also numerical simulation results were performed and compared to study the effect of the temperature dependent physical properties.

NOMENCLATURE

A Total area (m^2)
A_c Flow area (m^2)
c_p Specific heat at constant pressure ($J\ kg^{-1}\ K^{-1}$)
D_h Hydraulic diameter (m)
f Fanning friction factor
h Channel height (mm)
j Colburn factor
k Turbulent kinetic energy, $m^2 \cdot s^{-2}$
l Fin length in (mm)
m Mass flow rate (kg/s)
p Static pressure (Pa)
P_x Pitch in x-direction (mm)
P_y Pitch in y-direction (mm)
Pr Prandtl number
Q Thermal power (MW)
Re Reynolds number
t Fin thickness (mm)
T Static temperature (K)
T_{Ci} Cold side inlet temperature (K)
T_{Co} Cold side outlet temperature (K)
T_{Hi} Hot side inlet temperature (K)
T_{Ho} Hot side outlet temperature (K)
u_h Average velocity (m/s)
w Channel width (mm)
δ_{ij} Kronecker delta

μ Dynamic viscosity (kg m^{-1} s^{-1})
ν Kinematic viscosity (m^2 s^{-1})
ρ Density (kg m^{-3})
ω Specific dissipation rate, s^{-1}
Γ_k Effective diffusivity of k, kg·m^{-1}·s^{-1}
Γ_ω Effective diffusivity of ω, kg·m^{-1}·s^{-1}
G_k Generation of k due to mean velocity gradients, kg·m^{-1}·s^{-3}
G_ω Generation of ω due to mean velocity gradients, kg·m^{-3}·s^{-2}
Y_k Dissipation of k due to turbulence, kg·m^{-1}·s^{-3}
Y_ω Dissipation of ω due to turbulence, kg·m^{-3}·s^{-2}

INTRODUCTION

With the inevitable depletion of fossil fuels the need for hydrogen as a fuel storage medium in the future has been identified. Hydrogen can prove to be an attractive energy carrier if it can be demonstrated that it can be produced cleanly and in a cost-effective manner. Nuclear energy can be used as an abundant source of energy for high temperature processes, (up to 1000 °C) for production of hydrogen. The Sulfur iodine (S-I) Cycle, a baseline candidate thermo-chemical process consists of three chemical reactions that result in the dissociation of water. These reactions are as follows:

$I_2 + SO_2 + 2H_2O \rightarrow 2HI + H_2SO_4$ (120°C min.)
$H_2SO_4 \rightarrow H_2O + SO_2 + ½ O_2$ (850°C min.)
$2HI \rightarrow H_2 + I_2$ (450°C min.)

$H_2O \rightarrow H_2 + ½ O_2$

Theoretically, only water and heat need to be added to the cycle. From the above chemical reactions one can see that the splitting of the water molecule by this method requires a temperature of at least 850°C. All of the reactants, other than water, are regenerated and recycled. Figure 1 shows a concept for driving the S-I process using process heat from a modular helium reactor (MHR). The intermediate heat exchanger (IHX) would consist of heat-exchanger modules housed within a vessel, along with the primary coolant circulator. Alternatively, the intermediate heat transfer fluid could be a high-temperature, low-pressure liquid-salt, but this depends on tradeoffs between pumping power, heat exchanger mechanical design, and materials performance, cost, and safety.

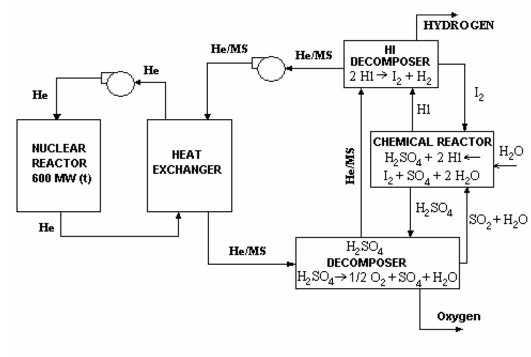

Figure 1. Schematic diagram of the plant concept

The present study considers an offset strip-fin type compact high temperature heat exchanger (shown in Figure 2), made of liquid silicon impregnated carbon composite. The ceramic composite material (CMC) is manufactured by impregnating the silicon into the pores of the carbon composites. The prototype heat exchanger is designed to operate at a thermal capacity of 50 MW, which is calculated based upon a general form of the energy balance equation.

$$Q = m \cdot C_p \cdot \Delta T \quad (1)$$

The operating conditions are shown in Table 1.

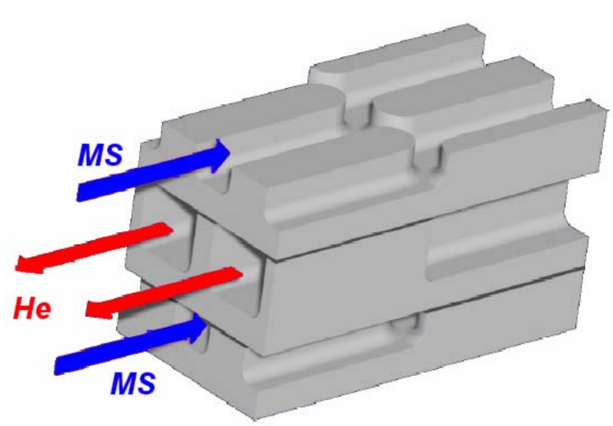

Figure 2. 3-D section of the flow channels

Table 1. Heat Exchanger Operating Conditions

Primary/intermediate fluids	Helium/Helium	Helium/ Liquid salt
Primary/intermediate pressures (MPa)	7.0/7.0	7.0/0.1
Primary inlet/outlet temperatures (°C)	1000/632	1000/632
Cold side inlet/outlet temperatures (°C)	560/975	560/975

Compact heat exchangers are used in a wide variety of applications such as HVAC and automobile systems. The need for lightweight, space saving, and economical heat exchangers has driven the development of compact heat exchangers. Compact heat exchangers are characterized by extended surfaces with large surface area/volume ratios that are often configured in either plate fin or tube fin arrangements. In a plate-fin heat exchanger, which finds diverse applications, a variety of augmented surfaces are used: plain fins, wavy fins, pin fins, strip fins, and perforated fins. However, offset strip-fins are widely used because of their compactness and periodic interruption of the boundary layers (Manglik and Bergles 1995). This means that the flow surface is arrayed with many strip-fins in a staggered fashion along the flow direction.

There have been considerable efforts in the past few decades to understand the flow field and heat transfer mechanisms of offset strip-fin heat exchangers. The first idea of forming some kind of empirical correlations was performed by Weiting (1975) and Shah et al. (1968). However, most theoretical solutions developed were based upon negligible fin thickness, but a few researchers, such as Patankar and Prakash (1981), compared experimental data with the information obtained from numerical simulations. According to the results obtained from their numerical computations the effect of fin thickness has a considerable effect on the pressure drop, rather than the heat transfer. The wall temperatures, which effect heat transfer, were set in such a way that they vary along the flow direction. Patankar and Prakash (1981) also compared results of local heat transfer and pressure drop across the channel with experimental data.

Kays and London (1964) conducted experiments of different configurations of offset strip-fin heat exchangers, and most of the theoretical correlations derived were based on comparison with these experimental results.

Joshi and Webb (1987) represented analytical models in order to predict the friction factor and the heat transfer coefficient. They were successful in analytically defining the laminar and turbulent regimes, but used numerical solutions to calculate Nusselt numbers in the laminar region. Flow visualization experiments were also performed, and their model was found to predict data within a range of 20 percent.

Kelkar and Patankar (1989) performed a three-dimensional computational study of constant property, steady laminar flow and heat transfer in the channels of rectangular offset-fin heat exchanger with no gap in flow direction. They assumed the flow to be periodically fully developed after a certain entrance length and hence concluded that the friction factor and heat transfer data can be made use in designing the whole apparatus without any significant error. A parametric study was made for various values of aspect ratio and fin-length parameter and the results were compared to experimental data.

Manglik and Bergles (1995) were the first to come up with an empirical model which can be used to calculate the friction factor and heat transfer coefficient for all three flow regimes, namely the laminar, transition, and turbulent regions.

DeJong et al., (1998) perfomed detailed experimental and numerical analysis in similar offset strip-fin geometries presented. For the numerical simulation they approximated the array as a periodic repetition of a basic unit. Thus, it has been assumed that the flow is both hydrodynamically and thermally fully developed in the fin array, and the effects of entrance and exit have been neglected. They concluded that the importance of the analysis of unsteady nature of the flow.

Saha and Acharya (2004) conducted a numerical study to analyze the unsteady three-dimensional flow and conjugate heat transfer heat transfer in a channel with inline and staggered arrays of periodically mounted square posts. They considered one periodic module to perform the numerical solution and showed the importance of conjugate heat transfer. Their results show that for the staggered fin arrays the flow may become unsteady at Re = 400.

Most of the research that has been done in the field of offset strip-fin compact heat exchangers has used rectangular fins with sharp edges and no pitch in the flow direction. However, the present study considers an offset strip-fin heat exchanger with a gap between adjacent fin rows in the flow direction. This study uses numerical solutions in order to analyze the effect of curved fin edges on heat transfer and pressure drop. Of specific interest is the level of enhancement by two different flow geometries, and the evaluation of the thermal performance of the prototype heat exchanger design.

CFD CALCULATIONS

The properties of the flow medium in a heat exchanger vary according to changes in temperature and pressure. For a gas with a small Mach number the pressure can be considered as being independent of density; therefore, in the present study the fluids are treated as being incompressible. But the density, as well as viscosity and thermal conductivity, are dependent on temperature. The computational effort for a variable property fluid is larger than a constant property fluid, but computations with temperature-dependent properties varied by less than 10 percent when compared to the same runs with constant fluid properties.

Governing Equations

For the convenience of utilizing a tensor notation the x,y,z coordinates are denoted as x_1, x_2, x_3 and the x, y, z components of velocity as u_1, u_2, u_3. Neglecting body forces the continuity, momentum, and energy equations can be written in Cartesian tensor form as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad (2)$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] \qquad (3)$$

where i = 1, 2, 3

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}(u_i(\rho E + p)) = \frac{\partial}{\partial x_i}\left(k\frac{\partial T}{\partial x_i}\right) \qquad (4)$$

where E is the total energy.

The momentum equation written in the above form is known as the Navier-Stokes equation. This set of equations is a general set, and along with some additional model equations they can be used for the calculation of any Newtonian viscous fluid flow process in Cartesian coordinates.

The turbulence modeling was performed using the standard k-omega model, an inbuilt module in the commercial code FLUENT (2003). The turbulence kinetic energy k, and the specific heat dissipation rate ω, is obtained from the following transport equations.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k - Y_k + S_k \qquad (5)$$

And

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + S_\omega \qquad (6)$$

Temperature Dependent Physical Properties

When there is a large temperature difference between the fluid and the surface the assumption of constant fluid transport properties may cause some errors, because the transport properties of most fluids vary with temperature. These property variations will then cause a variation of velocity and temperature throughout the boundary layer or over the flow cross section of the duct.

For most liquids, such as the liquid-salt FLINAK, the specific heat, thermal conductivity, and density are nearly independent of temperature, but the viscosity decreases markedly with increasing temperature. It is also important to note that the Prandtl number of liquids also varies with temperature, similar to that of viscosity.

In the case of gases, like helium, the density, thermal conductivity, and viscosity all vary at the same rate with respect to temperature. The specific heat varies only slightly with temperature, and the Prandtl number does not vary significantly, which was shown by Kakac et al. (1987).

Hence, in order to study the influence of temperature dependent physical properties on the numerical simulations simple polynomial equations were formed. With those equations having the physical properties defined only as a function of temperature.

3-D Numerical Simulation for Full Channel Length

For the heat transfer simulations the present study performed numerical simulation for the full channel length of 0.9 m. A journal file generating code was written using PASCAL, which creates the journal file that can be run in GAMBIT, and GAMBIT is used to generate the 3-D computational domain for the 37 modules of the heat exchanger. The computational geometry was created according to Figure 3 and Table 2. The mesh file was created with about one million nodes, which FLUENT uses for hydrodynamic and heat transfer simulations. Normal velocity inlet and pressure outlet boundary conditions were used at the entrance and exit of the channel, respectively. The coupled heat transfer boundary conditions were used to solve the energy equation. Numerical computations were performed for both rectangular and curved fin edges.

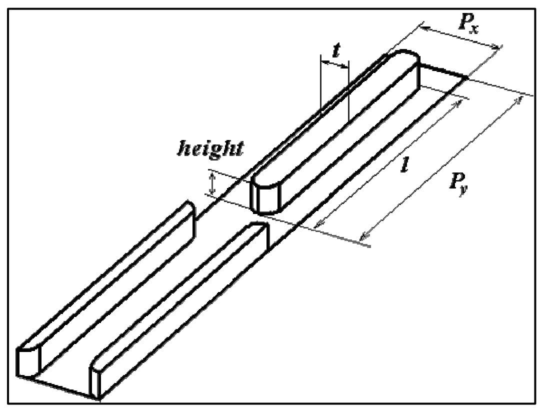

Figure 3. 3-D section of a single flow channel with gap length showing the dimensional parameters

Table 2. Heat Exchanger Channel Dimensions

Geometric parameters	Helium side (mm)	Liquid salt side (mm)
Fin length (l)	10	10
Channel height (h)	2	1
Fin thickness (t)	0.75	1.25
Pitch in flow direction (P_y)	12	12
Pitch in span wise direction (P_x)	3	3

Offset Strip-Fin Channel Dimensions

Figure 3 shows the three-dimensional section of the schematic geometry that was considered in the present study. Each fin has length (l), thickness (t), height (h), pitch in x-direction (P_x) and pitch in y-direction (P_y). Table 2 summarizes the flow channel dimensions for the baseline heat exchanger design. The dimensions that were chosen for the baseline heat exchanger geometry were based upon some initial sensibility studies that were performed for the thermal design. For the baseline heat exchanger design there is a gap length present in the flow direction between the fins i.e., there is some distance between the trailing edge of the fins of one row to the leading edge of the fins in the downstream row. The gap is defined as the difference between the pitch in the flow direction and the fin length, i.e., *Gap length* = $P_y - l$.

Computational Method

The solution of the velocity field was accomplished using the SIMPLE algorithm, which resulted in a faster convergence of the iterations. All computations were carried out with approximately one million nodes, and computations were done for the baseline heat exchanger design. Convection to the fin surface and heat conduction through the solid was considered in the SIMPLE algorithm, and the flow through the channels, between the fins, was considered as incompressible with both laminar and turbulent flow models.

Boundary Conditions

The inlet temperature, inlet velocity (based on the mass flow rate), and the exit pressure were used as boundary conditions. Symmetric boundary conditions were used in the span wise direction of the flow channels and along the channel height, due to the symmetrical nature of the heat exchanger geometry. At the symmetric planes heat flux is assumed to be zero, and the normal velocity component at the symmetry plane is also zero; therefore, no convective flux across that symmetry plane occurs. Thus, the temperature gradients and tangential components of the velocity gradients in the normal direction are set to zero.

Conjugate heat transfer, which includes conduction through the material and convection through the fluids, was used in order to solve the energy equation. No other thermal boundary conditions were required for the problem since the solver will calculate heat transfer directly from the solution in the adjacent cells. In order to use the Conjugate Heat Transfer (CHT) option in FLUENT the capability of FLUENT to accurately simulate CHT must be validated. One of the ways of validating heat transfer results is to compare results with analytical solutions, and this was done for the Nusselts numbers of laminar flow through circular, rectangular, and infinite width (flat plate) channels.

Solution Algorithm

In the present study a general curvilinear coordinate grid generation system is used to discretize the computational domain into a finite number of control volumes. With proper control of the grid density the computational domain can be considered to have two main regions as shown in figure 4. In the first region the finer mesh sizes are prepared near the fin wall to resolve the secondary flows, vortices and flow separations, where high gradients are expected, and in the second region the coarse mesh sizes are selected for the case where the flow is relatively uniform. The first order upwind numerical scheme and SIMPLE algorithm in FLUENT (2003) is used to discretize the governing equations.

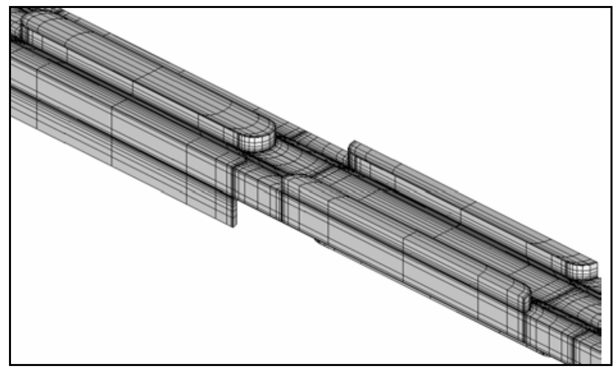

Figure 4. 3-D Mesh for the helium channel

RESULTS AND DISCUSSIONS

The simulations were performed for a helium-liquid salt heat exchanger as described above. The overall hydrodynamic and thermal performance of the two fin types (curved and rectangular) candidate heat exchanger models were calculated and compared using constant material properties. Additionally, the analytical calculations using the empirical correlations by Manglik and Bergles (1995) were performed to predict and compare CFD results. It was found that for both types of fin cases the helium side had a larger pressure drop compared to that of the liquid salt side, due to the higher velocity of the gas. Therefore, less pumping power is required for the liquid salt side in spite of having a reduced flow area for providing the same heat transfer performance. The vectors plots for the helium side indicated the presence of recirculation regions, which are local hotspots that can degrade thermal performance. Parametric studies with one module were used to investigate the influence of the Reynolds number on the length of the recirculation zone ("reattachment" length). As the Reynolds number increased the reattachment length and the magnitude of the vortices increased and had a detrimental effect in increasing the pressure drop along the flow channel. Similar studies were not performed for the liquid salt side, because of the low Reynolds number flow occurring in the flow channel.

Tables 3 and 4 show the CFD results for the two cases of the offset strip-fin heat exchanger geometries with constant material properties and variable material properties.

Table 3. Heat Exchanger Performance from CFD Calculations (Curved Fin Edge Case)

Property	Constant material properties	Variable material properties	% Difference
Helium side Friction Factor (*f*)	0.02386	0.02386	0
Liquid Salt side Friction Factor (*f*)	0.10607	0.11636	10
LMTD (K)	39	39	0
Thermal Capacity (MW)	50.8	50.9	0.2

Table 4. Heat Exchanger Performance from CFD Calculations (Rectangular Fin Edge Case)

Property	Constant material properties	Variable material properties	% Difference
Helium side Friction Factor (*f*)	0.02666	0.02652	0.5
Liquid Salt side Friction Factor (*f*)	0.11500	0.12583	9
LMTD (K)	39	39	0
Thermal Capacity (MW)	50.8	50.9	0.2

The log mean temperature difference (LMTD) is defined as the temperature difference at one end minus the temperature difference at the other end of the heat exchanger divided by the natural logarithm of the ratio of these two temperature differences, i.e.:

$$LMTD(K) = \frac{\Delta T_0 - \Delta T_i}{\ln\left(\frac{\Delta T_0}{\Delta T_i}\right)} \quad (7)$$

where,

$$\Delta T_0 = T_{Ho} - T_{Ci}(K)$$
$$\Delta T_i = T_{Hi} - T_{Co}(K)$$

The above definition of LMTD involves two assumptions: (1) the fluid specific heats do not vary significantly with respect to temperature, and (2) the convective heat transfer coefficients are relatively constant throughout the heat exchanger.

Figures 5 and 6 show the temperature change across the flow channels of the heat exchanger as predicted by the CFD results.

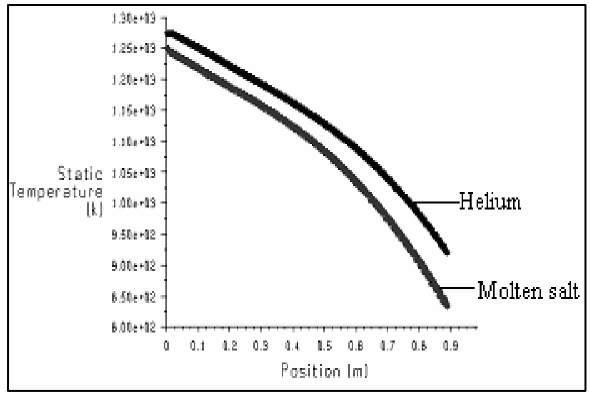

Figure 5. Temperature profile along the flow channels with constant material properties

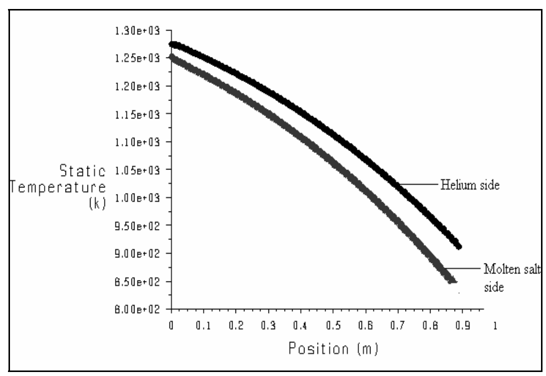

Figure 6. Temperature profile along the flow channels with variable material properties

Helium side

The CFD simulations for the hydrodynamics predicted almost no variation in the friction factor values for both heat exchanger channel types with constant and temperature-dependent properties. CFD results also showed insignificant difference in thermal performances for the case between constant and temperature-dependent material properties. Hence, taking into account the computational time for turbulence modeling and the effect of a variable physical properties model, it was decided to perform turbulence modeling with a constant physical properties model. Figure 7 shows a sample of temperature contours and velocity vectors in the middle of helium channel.

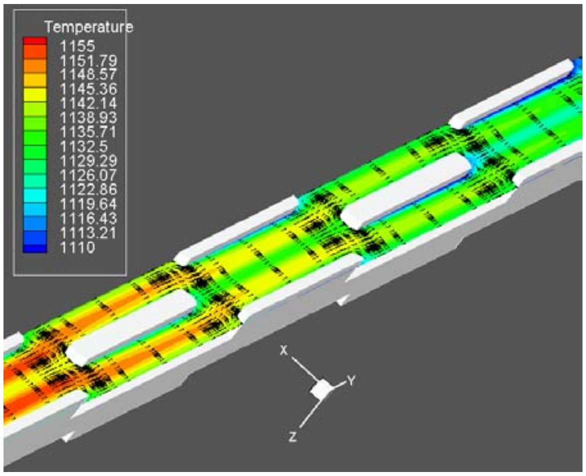

Figure 7. Velocity vectors (m/s) and temperature (K) contours in the middle of the helium channel

The turbulence modeling was performed for the helium channel using the K-ω turbulence model. The turbulence intensity was taken to be 1% since the helium channel flow is present in the lower transition flow regime (Re=2400).

Table 5. Helium side hydrodynamic performance comparison from CFD Calculations

Property	**Curved Fin Edge case**	**Rectangular Fin Edge Case**	**% Difference**
Helium side Friction Factor (*f*)	0.02409	0.03279	36

But a 36% difference in the helium side friction factor values predicted by the CFD as shown in table 5 for the curved versus rectangular fins was observed. This can be attributed to the flow constrictions caused by the sharp edges of the rectangular fins, and the reduced area of flow separation in the curved fin edge cases. However, the thermal capacities for both the rectangular and curved fin edge cases were predicted to be 51 MW; thus, indicating that the effect of the shape of the fin edges had negligible affect on the heat transfer.

Table 6. Helium side Friction Factor (*f*) results comparison between laminar and turbulent model CFD Calculations

Heat Exchanger Channel	**Laminar Case**	**Turbulent Case**	**% Difference**
Rectangular Fin Edge Case	0.02660	0.03279	36
Curved Fin Edge Case	0.02386	0.02409	1

Table 6 provides a comparison about the influence of the different numerical models used. It can be seen that the effect of turbulence is significant only for the friction factor values of the rectangular fin edge helium channel. Thus showing the more pronounced effects of blockage and flow disturbances in the rectangular fin edge channel.

Liquid salt side

The CFD simulations predicted approximately about 10% and 9% difference in the friction factor values between the curved fin edge and rectangular fin edge case with constant and temperature-dependent properties, respectively. The CFD results predicted the same thermal capacity (50 MW) for the liquid salt side for both constant and temperature-dependent properties and curved and rectangular fins. Thus, accounting for temperature dependent properties results in about a 10% difference in the pressure drop but no affect on the heat transfer. Figure 8 shows a sample of temperature contours and velocity vectors in the middle of liquid salt channel.

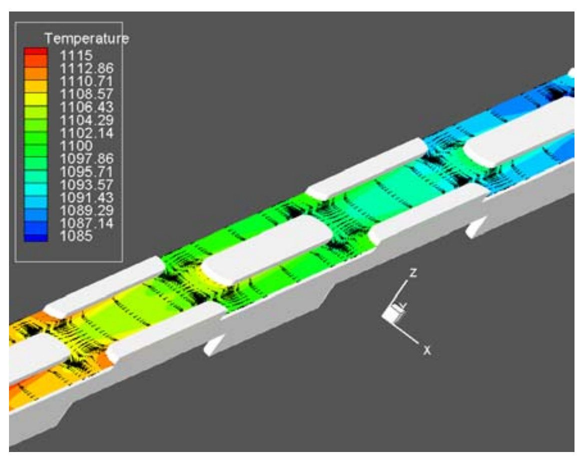

Figure 8. Velocity vectors (m/s) and temperature (K) contours in the middle of the liquid salt channel

Table 7. Liquid Salt side hydrodynamic performance comparison from CFD Calculations

Property	Curved Fin Edge case	Rectangular Fin Edge Case	% Difference
Liquid Salt side Friction Factor (*f*)	0.10607	0.11555	9

The impact of the curved versus rectangular fins resulted in a 9% difference in the friction factor values as shown in table 7 and no impact on the heat transfer performance.

The highest Reynolds number for the helium side used in the computations was 2400, which is in the lower transition region. Reynolds number is defined as

$$Re = \frac{U_h D_h}{\upsilon} \tag{8}$$

where,

D_h is the hydraulic diameter given by $D_h = \frac{4A_c}{A/l}$

At theses values of Re the flow is expected to be mostly laminar; although, it is possible that transition to turbulence may occur somewhat before Re = 2000. Also the flow may display instabilities and vortex-shedding from the trailing edges of the plates.

CONCLUSION

A three dimensional computational model was developed for the fluid flow and heat transfer in a compact off-set strip fin high temperature heat exchanger. The heat exchanger is an integral part of the interface between the nuclear plant and the hydrogen production plant. The manufacturing of the ceramic HTHX using liquid silicon impregnated carbon-carbon composites allows for flexibility in producing the desired geometry. The model is based on solving a set of incompressible momentum and energy equations over 37 periodic modules (0.9 m total length) of the heat exchanger. The flow field is affected by blockage and recirculation zones caused by sharp rectangular edges and narrow gaps. The results show that the CFD tools such as FLUENT can adequately demonstrate the flow physics and heat transfer for these types of complex geometries and is also useful for fin optimization studies. As Saha and Acharya (2004) found out the unsteady nature of flow for these heat exchanger channels, the inclusion of flow unsteadiness into the numerical model is also a future scope. Also, since the results published are purely from CFD studies only comparison with experimental results will provide a more realistic idea about design and performance, which is slated for future work. It was observed that the heat exchanger channels with curved fin edges yield a better overall performance by lowering the pumping power. The temperature dependent physical properties had some effect on the liquid-salt channel flow, but its effect on the overall performance was found to be insignificant. The effect of temperature dependent physical properties can be neglected in future turbulence studies taking into consideration the additional amount of computational time.

ACKNOWLEDGMENTS

This work was funded by the US Department of Energy under the contract DE-FG04-01AL67356. The authors would also like to acknowledge Prof. Per F. Peterson and his research team at the University of California, Berkley for their helpful suggestions.

REFERENCES

FLUENT Inc., 2003, *FLUENT 6.1 User's Guide*, Lebanon, NH.

Manglik, R.M., and Bergles, A.E., 1995, "Heat Transfer and Pressure Drop Correlations for Rectangular offset strip fin compact heat exchanger", *Exp. Thermal and Fluid Science*, **10**, pp. 171-180.

Weiting, A.R., 1975, "Empirical Correlations for Heat Transfer and Flow Friction Characteristics of Rectangular offset Plate Fin Heat Exchangers", *J. of Heat Transfer*, 97, pp. 488-490

London, A.L., and Shah, R.K., 1968, "Offset Plate fin Surfaces- Heat Transfer and flow Friction Characteristics", *Transactions of ASME*, pp. 218-228.

Patankar, S.V., and Prakash, C., 1981, "An Analysis of the Effect of Plate Thickness on Laminar Flow and Heat Transfer in Interrupted-Plate Passages," *Int. J. Heat Mass Transfer*, **24**, pp. 1801-1810.

Kays, W.M., and London, A.L., 1964, "Compact Heat Exchangers", 2nd ed., McGraw Hill, New York

Joshi, H.M., and Webb, R.L., 1987, "Heat Transfer and Friction in offset strip-fin heat exchanger", *Int. J. Heat Mass Transfer,* **30**, pp. 69-84.

Kakac, S., Shah, R.K., and Aung, W., 1987, "Handbook of Single-Phase Convective Heat Transfer", John Wiley & Sons, Inc.

Kelkar, K.M and Patankar, S.V., 1989, "Numerical Prediction of Heat Transfer and Fluid Flow in Rectangular offset-fin arrays", *Numerical Heat Transfer, Part A*, **15**, pp 149-164.

Saha, A.K and Acharya, S., 2004, "Unsteady Flow and Heat Transfer in Parallel-Plate Heat exchangers with In-line and Staggered arrays of Posts", *Numerical Heat Transfer, Part A*, **45**, pp 101-133.

DeJong, N.C. et al., 1998, "A Complementary Experimental and Numerical Study of the Flow and Heat Transfer in Offset Strip-Fin Heat exchangers", *Transactions of ASME*, **120**, pp 690-698