Optimization of the Geometry of a Heat Sink

22 April 2007

Matthew Bronson de Stadler

Table of Contents for Final Binder

Optimization of the Geometry of a Heat Sink

A Thesis
in STS 402

Presented to

The Faculty of the
School of Engineering and Applied Science
University of Virginia

In Partial Fulfillment
of the Requirements for the Degree

Bachelor of Science in Aerospace Engineering

by

Matthew Bronson de Stadler

27 March 2007

On my honor as a University student, on this assignment I have neither given nor received unauthorized aid as defined by the Honor Guidelines for Papers in Science, Technology, and Society Courses.

____

Approved ____

Technical Advisor – Hossein Haj-Hariri

Approved ____

Science, Technology and Society Advisor – Ingrid H. Townsend

Acknowledgments

This work was supported by the Virginia Space Grant Consortium through an Undergraduate Aerospace Research Scholarship. The author would like to thank Professor Hossein Haj-Hariri, his technical advisor at the University of Virginia, for help in this undertaking. He would also like to thank Professor Ingrid Townsend, his STS advisor, for all her help. Lorenzo Valdevit provided a copy of his group's paper on skin cooling for scramjets which proved to be very helpful in determining the thermal network presented in the report.

Table of Contents

List of Figures

List of Tables

Abstract

Cellular materials have properties similar to conventional materials on the macroscopic level but possess several advantages at the microscopic level, including the ability to be manufactured down to a very small scale. With this ability, new geometric configurations for a heat sink are able to be considered. This paper details the results of a study to develop a geometry based optimization tool for heat sink design.

With the principle of superposition, the analysis of a heat sink can be simplified by using a repeating cell. For the repeating cell, the following questions are posed: Given a certain percentage of the volume available for channels for fluid to pass through, where should one place the channels? What shape should they be? To answer these questions a theoretical approach was used with testing and analysis performed using computational fluid dynamics software. Three unsuccessful schemes are presented, with explanations as to why they did not prove successful. Lessons learned from these attempts are applied to the ongoing development of an optimal heat sink for use in skin cooling of a hypersonic vehicle.

Nomenclature

A	=	area
c_p	=	specific heat capacity
h	=	convection heat transfer coefficient
H	=	height
k	=	thermal conductivity
L	=	length
$\dot{m}$	=	mass flow rate
n	=	number of channels
ρ	=	density
p	=	pressure
P	=	power
Q	=	heat flux
$\dot{Q}$	=	heat flux rate
s	=	perimeter
μ	=	dynamic viscosity
T	=	temperature
V	=	velocity
W	=	width

Chapter I. Introduction

This chapter presents the scope and motivation for this senior thesis project. A roadmap for the rest of the paper is given at the end of the chapter.

Heat sink research and development has had a long history which is still ongoing with efforts to improve design and performance.* Incropera and DeWitt (1996) list enhanced heat transfer surfaces as an important contemporary research area. "With heightened concern for energy conservation, there has been a steady and substantial increase in activity. A focal point of this work has been heat transfer enhancement, which includes the search for special heat exchanger surfaces through which enhancement may be achieved (p. 618)." Recent developments in cellular materials allow for the consideration of designs previously not possible. Cellular materials allow for the construction of very small heat sinks, with passageways for fluid to pass through on the order of several millimeters thick as shown in Figure 1 (Gibson and Ashby 1997).

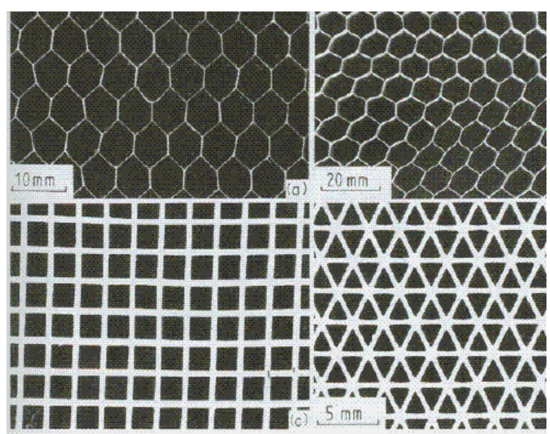


Figure 1. Cellular structures.
Gibson, L.J., & Ashby, M.F. (1997). *Cellular Solids: Structure and properties*.

Their superior properties, when compared with conventional materials, make cellular materials very desirable for a wide range of applications where size, weight, and

---

*As heat sinks and heat exchangers are very similar devices, occasionally in this paper discussion of heat exchangers will be presented with the intention that the results are applicable to heat sinks as well.

efficiency are important. The decision to investigate optimal geometries for heat sinks was inspired by these recent developments. Optimal geometries are characterized by enhanced heat transfer surfaces. These allow devices to take advantage of one of the following options: size reduction, increased thermodynamic process efficiency which leads to lower operating costs, increased heat exchange rate for fixed fluid inlet temperatures, or reduced pumping power for fixed heat duty (Webb and Kim 2005 p. 2).

To the best of the author's knowledge, a study of optimal geometries for a fixed volume heat sink with a fixed percentage of the volume available for internal cooling channels has not been conducted. The goal for this paper was to develop a methodology which would allow one to develop an optimal heat sink based on geometrical considerations independent of working fluid choice and heat sink material choice. A general solution was sought so that the results obtained could be used for heat sink applications ranging from small-scale electronics cooling to temperature regulation in large energy facilities.

As heat sinks tend to be very complex, the analysis was simplified by considering designs employing a repeating cell. Using the principle of superposition, the characteristics of the entire heat sink may be determined from the characteristics of a representative cell. For a fixed volume cell with a given percentage of the volume available for cooling channels, two questions were posed:

i. How should one layout the channels?
ii. What shape should the channels be?

A purely theoretical approach was employed to answer these questions, with testing and analysis performed using computational fluid dynamics (CFD) software. Several attempts were made to develop a metric for comparing designs with different features to objectively determine the best one. Each attempt is presented with explanation as to why it was not successful. Current work being done on skin cooling for a hypersonic vehicle is also presented.

The structure of this paper is as follows. Chapter II contains a literature review. The social and ethical implications associated with this research are given in Chapter III. Chapter IV presents the problem statement. 2-D conduction modeling efforts are given in Chapter V. Chapter VI details an attempt to generate performance trends. Attempts to develop a performance metric are given in Chapter VII. Skin cooling of a hypersonic vehicle, an application of this general research, is presented in Chapter VIII. Chapter IX contains conclusions and recommendations for future work.

Chapter II. Literature Review

This chapter provides background information from relevant literature for this senior thesis project. Heat sinks and heat exchangers, cooling pipes, optimization techniques, cellular materials, and air and space vehicles are all covered.

A. Early History

Shah (1978) offers a substantial treatment of the history of heat exchanger development up to 1975 (p. 2). A brief summary will be given here. The study of heat transfer in laminar flow through a closed conduit was first made by Graetz in 1883, and later independently by Nusselt in 1910. Drew prepared a compilation of existing theoretical results for heat transfer data in 1931. Dryden et al. in 1932 compiled fully developed laminar flow solutions for ducts of various geometries. In 1964 Kays and London published a compilation containing data pertinent to compact heat exchangers. With the help of 30 British industries, Porter compiled the laminar flow solutions for Newtonian and non-Newtonian liquids with constant and variable fluid properties in 1971. The heat transfer literature up to 1967 was reviewed by Kays and Perkins who provided available results in terms of equations, tables, and graphs for design purposes.

B. Technical Background

A heat sink is a device that removes heat from an object. A closely related technology is a heat exchanger. A heat exchanger is a device which transfers thermal energy, heat, from a hot source to a colder one. This is usually done with two working fluids.

The ability to regulate temperature makes heat sinks suitable for a wide variety of applications, including nuclear power plants, satellites, air conditioners, and electronics.

This versatility makes heat sinks a very valuable technology for the future. It also causes difficulty for design, as described by Afimiwala, Mayne and Shah (1978). “The heat exchanger design problem is intricate because it involves many variables for geometry and operating conditions, also heat exchangers are designed for a variety of applications having different objectives (p. 185).” This complexity makes heat sink design a difficult optimization problem.

C. Heat Sinks and Heat Exchangers

Heat sink research and development has had a long history which is still ongoing with efforts to improve design and performance. Incropera and DeWitt (1996) list enhanced heat transfer surfaces as an important contemporary research area. “With heightened concern for energy conservation, there has been a steady and substantial increase in activity. A focal point of this work has been *heat transfer enhancement*, which includes the search for special heat exchanger surfaces through which enhancement may be achieved (p. 618).” Webb and Kim (2005) discuss the benefits of enhanced heat transfer surfaces; they allow one to take advantage of one of the following options: size reduction, lower operating costs, increased heat exchange rate, or reduced pumping power (p. 2).

Enhancement techniques are separated into two areas, passive and active techniques. Passive techniques involve special surface geometries or fluid additives for enhancement while active techniques require external power such as electric or acoustic fields and surface vibration (Webb and Kim 2005). While many of these techniques have great theoretical promise, it is important to remember that a given enhancement geometry might not be able to be manufactured for all desired materials.

Compact heat exchangers are becoming increasingly more common in applications with limited size and weight availability. These operate like any other heat exchanger but are characterized by a very large heat transfer surface area per unit volume. This allows for greater cooling to take place in a smaller space. Examples of compact heat exchangers include car radiators and the human lung (Cengel 2007).

D. Cooling Channels

In the past, difficulty in manufacturing confined most heat sinks to using straight channels with a circular or square cross-section for fluid flow. In the 1960s and 1970s, significant work was done to determine characteristics of non-circular ducts. Much of this data has been summarized by Kays and London (1964, 1978) and related literature. More recently there has been significant work done in the area of helical tubes (Acharya, Sen and Chang 2001). These tubes can have any cross-section, most often a circular one is chosen, and twist to form coils around either one or two axes. The number of coils can vary from one, a very gentle curve, to an almost infinite number of tangential curves like wires on a solenoid.

The advantage of incorporating coiled tubes is that they increase fluid mixing which increases the heat transfer rate. The disadvantage of these designs is that they also increase the pumping power required. Intuition leads one to believe there is an optimal way to employ these twisted tubes in a heat sink in order to get maximum mixing with minimum pressure increase. This is one of the key items to be investigated in the future.

E. Optimization Techniques

Optimization techniques are very important as well since there are so many variables and design options. Many possibilities exist for comparing the performance of different heat transfer surfaces. Shah (1978) lists several (p. 196), categorizing them as

(i) comparison based on the Colburn heat transfer modulus and the apparent mean Fanning friction factor

(ii) comparison of heat transfer as a function of fluid pumping power

(iii) miscellaneous direct comparison methods

(iv) performance comparisons with a reference surface

These techniques are good for single surface selection but do not necessarily result in an optimal heat sink. Another option is described by Afimiwala et al. (1978); one selects a single performance measure and defines it quantitatively seeking to minimize or maximize it, making it an 'objective function.' The customer then specifies performance constraints which results in the basic optimization problem being formulated as follows:

Minimize $f(\underline{x})$
Subject to $g_i(\underline{x}) \leq 0$ for $i = 1, 2, \ldots, p$
And $h_j(\underline{x}) = 0$ for $j = 1, 2, \ldots, q$

The functions may be either linear or non-linear. The vector $\underline{x}$ contains n unknowns which are to be adjusted to minimize or maximize the objective function f(x) (p. 185, 186).

A third optimization technique available is the Taguchi method. This method was developed in Japan to increase the quality of manufactured goods and has since been adapted for many other uses. The Taguchi method is a statistical technique which determines the variables most critical to a given design. When used properly, it determines the importance of the interaction between two variables to the final answer. It does not tell how they interact but ultimately it allows the experimenter to save substantial time in identifying the parameters most relevant to the solution of the problem.

The most practical advantage of the Taguchi method is that it will reduce the number of computer simulations necessary for analysis, which is important as CFD simulations tend to be very time consuming. Pluntze and Eberhardt (1998) were successful in applying the Taguchi method to a CFD study to extend results from incompressible wind tunnel data at moderate Reynolds numbers to flight Reynolds number values. Past studies such as this involving both optimization and computational tools are expected to prove useful in setting up an optimization scheme for this project.

F. Cellular Materials

Developments in cellular materials allow for consideration of significantly more complex heat sink designs. Cellular solids, a subcategory of cellular materials, are very relevant to this project. Gibson and Ashby (1997) define a cellular solid as “One made up of an interconnected network of solid struts or plates which form the edges and faces of cells (p. 2).” They offer a wide range of properties, making them applicable to such diverse applications as thermal insulation, packaging, structural support, and buoyant devices. Some advantages of cellular materials include a relatively low density, a highly porous nature, and the ability to be tailored down to a very small scale. Also, cellular materials perform like conventional materials on the macroscopic scale. The biggest advantage of cellular materials is that they allow one to construct a heat sink using the desired material, but with significantly more customizable geometries, especially on a very small scale (Gibson and Ashby 1997).

G. Air and Space Vehicles

Weight is incredibly important for launch vehicles. For example, Mangonon (1999) notes when discussing the space shuttle that it has been estimated that the savings per pound of weight in the orbiter vehicle structure or systems amounts to over $30,000 over the original predicted mission life of 10 years (p. 777). A more efficient heat sink will allow for size and weight reduction from current designs. This will allow the launch vehicle to operate at a lower mass or to carry an increased payload.

Providing skin cooling for flight vehicles is another potential application for a heat sink. As the maximum possible speed continues to increase, the thermal demands placed on the exterior of the flight vehicle also increase. To control these intense heat fluxes and extreme temperature values, one could develop a heat sink located just below the aircraft skin as an alternative to high-performance ceramics. This does significantly increase the complexity and additional support infrastructure will be required for such a cooling technique to be practical. A cost-benefit analysis should be done to determine whether such a design would ultimately be feasible.

Chapter III. Social and Ethical Implications

This chapter discusses the social and ethical implications of this research from improved energy efficiency to increased complexity.

Heat sinks are found in a wide variety of applications making it difficult to say what results are likely to occur from technological advances. Since they are often coupled with other devices, to fully understand their future impact, one must also understand the future impact of the systems they occur in. Given their widespread usage and interdependence with other devices, predicting the impact of improved heat sinks is very challenging.

The main benefit of this research is that it will continue the development of more efficient heat sinks. Hedderich, Kelleher and Vanderplaats (1981) state the importance of ensuring good design choices for a heat sink. “Studies have shown that a poor choice of either the heat transfer surfaces or design parameters can more than double the costs chargeable to a heat exchanger (p. 683).” A well designed heat sink will result in lower costs and allow for the improvement of many related systems. This will have many benefits for the world community, beginning with less expensive energy production.

This technology will most likely benefit the middle class and the wealthy. While these improved designs will save money in the long run, they are going to have large capital costs which are likely to dissuade many potential users, especially those with little disposable income. It will take some time before the benefits of cheaper energy production reach the lower class. However, once these improved heat sinks become standard, everyone will benefit greatly from the reduced cost of energy production. The military and power companies are likely to be the main beneficiaries as they have strong interest in the development of improved heat sinks since these devices are integral members in many of their critical systems.

While the basic idea behind a heat sink is simple, many dependent variables affect the performance, making heat sink design somewhat of an art. This leads to the development of designs which tend to be very complex. Given that fact, the final product of this research is likely to be even more complex, which will lead to a further gap between lay users and the knowledge elite responsible for heat sink design. This widening knowledge gap is a trend to be aware of for the future.

Presumably manufacturing technology will continue to improve so that more advanced designs can be built at standard factories. If not, new facilities will have to be developed to take advantage of this research. While this is likely to cost money, it will also create jobs to develop and maintain the new machinery.

Heat sinks are often members of a power cycle. These systems will improve as the components improve, and the improvement of the heat sink will lead to improvements in other components, for example, a better condenser. However, it is hard to ensure that the devices which employ these power cycles will be operating in a benevolent manner.

Ethical difficulties arise with this project in that heat sink can be found in many items from air conditioners to nuclear power plants. Problems associated with a more efficient air conditioner will be much less worrisome than those with a more efficient nuclear power plant. With a cheaper heat sink, and hence an improved power cycle in general, less efficient energy sources could see wider use. One possible scenario is that the number of coal driven power plants could increase, contributing substantially to air pollution.

Currently, the future behavior this research will enable can not be accurately estimated. However, after considering the social and ethical implications, the expected benefits far outweigh the possible negative results and this trend is likely to continue into the future. With an ever increasing emphasis on energy efficiency, it is important to ensure that devices operate in an optimal manner. This research supports that goal by looking to determine tools that will allow future engineers to develop optimal designs to best take advantage of the limited resources available.

Chapter IV. Problem Formulation

This chapter presents the problem statement that was investigated for this research. The framing of the problem is discussed along with information on how important parameters affect heat sink performance for a given design.

A. Framing the Problem

To determine an optimal design for a heat sink, the principle of superposition was used to reduce the design area to the smallest repeating cell as shown in Figure 2.

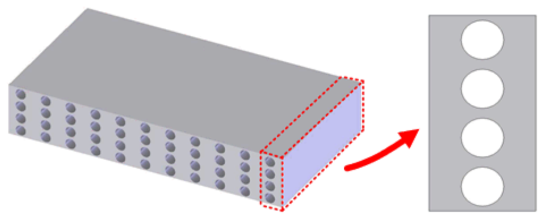


Figure 2. Repeating cell concept.

This allows one to determine the characteristics of the complex heat sink from a simple repeating cell. A fixed percentage of the cell volume is available for channels for fluid to pass through. Within the cell, a number of design variables are able to be considered including:

i. Number of channels
ii. Placement of channels within the normal face of the metallic block
iii. Channel shape (circle, square, triangle,…)
iv. Channel axial variation (straight versus coiled tube)
v. Fluid mass flow rate

In addition to passive enhancement techniques like those mentioned above, active enhancement techniques requiring external power can be used to improve heat transfer. Attention was focused on passive techniques due to the limited scope of this paper.

What does it mean for a design to be optimal? For the purpose of this study, an optimal design is one in which the greatest thermal protection is offered at the lowest pumping power. This thermal protection can be quantified using values like the maximum temperature, average temperature, or maximum heat flux. The optimal solution is believed to lie between two extreme cases, one large channel which does not remove very much heat but has a very low pumping power and infinitely many small channels which remove a very large amount of heat yet require extremely high pumping power. Another goal of this study was to ensure that the minimum amount of working fluid would be needed to provide the desired cooling.

The situation under consideration is that a device is subjected to a thermally demanding environment and a limited space for cooling is available. A heat sink consisting of a rectangular block with channels for cooling fluid to flow through is placed between the device and the hot source to offer steady-state thermal protection for the device. For this analysis the problem was modeled as shown in Figure 3(a).

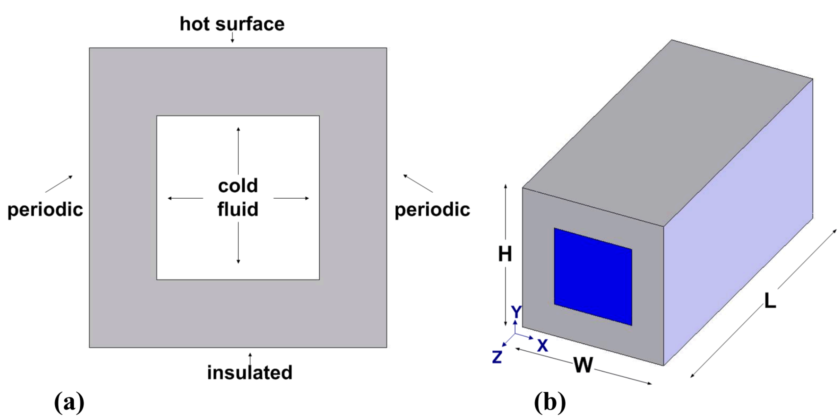


Figure 3. (a) Problem setup. (b) Model dimensions.

The top of the representative unit cell is exposed to a hot surface, a periodic boundary condition is applied to the sides due to symmetry, the bottom is assumed to be insulated, and cold fluid passes through the channels. A fixed volume is available for the heat sink as shown in Figure 3(b).

B. Design Considerations

Two modes of heat transfer occur in this heat sink: conduction and convection. Figure 4 shows conduction in the metal on the normal face of three different possible designs.

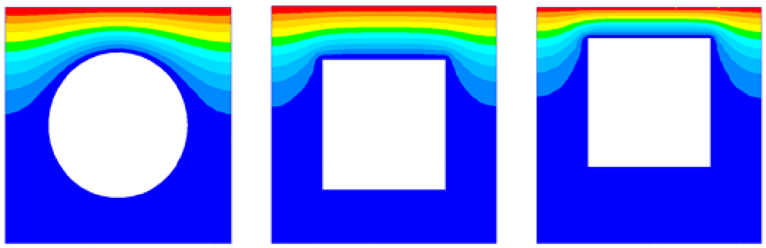


Figure 4. Temperature profile on the normal face for several geometries. Red is hot and blue is cold.

Heat is uniformly applied to the top surface and travels down toward the bottom. All of the heat is transferred from the block metal to the working fluid at the fluid-metal interface but depending on where the heat enters the cooling channel, the temperature profile of the metal block will vary. Figure 5 shows the temperature variation in the axial direction for three different mass flow rates through a square duct.

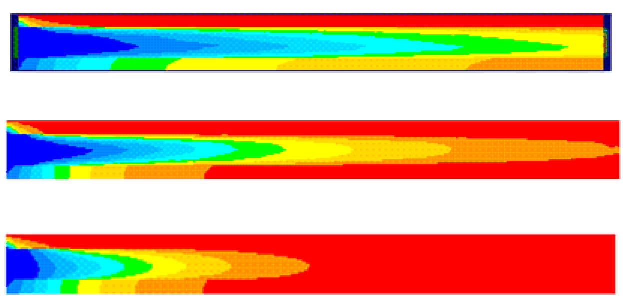


Figure 5. Temperature profile in the axial direction for flow in a square duct. Red is hot and blue is cold.

The mass flow rate of the top design in Figure 5 is too high and results in the fluid exiting the heat sink before it is fully heated. This is undesirable because excessive pumping power is required and the fluid is still able to remove additional heat when it exits the channel. The mass flow rate of the bottom design in Figure 5 is too low and as a result the fluid becomes fully heated well before it leaves the heat sink. In this case, the fluid stops doing effective thermal work before it leaves the channel. The middle design in Figure 5 is very close to the optimal design in that the fluid leaves the heat sink just as it becomes fully hot.

In addition to searching for optimal shapes and layouts for channels, knowledge of the variables governing the ideal mass flow rate is also desired. By applying the conservation of energy to a differential control volume for internal flow in a tube, one can obtain an expression for the mean temperature of the fluid (Incropera and DeWitt 1996). For pipes with a constant surface heat flux the following relationship is obtained:

$$T_m(x) = T_i + \frac{\dot{Q}_s s}{A_s \dot{m} c_p} x \tag{1}$$

Where $T_m(x)$ is the mean fluid temperature as a function of axial position, T_i is the fluid entrance temperature, $\dot{Q}_s$ is the surface heat flux, c_p is the specific heat capacity, s is the channel perimeter, $\dot{m}$ is the mass flow rate, and A_s is the exposed surface area. If instead of a constant surface heat flux, a constant surface temperature is used, one finds that

$$T_m(x) = T_s - (T_s - T_i) e^{\left(-\frac{s h_{avg}}{\dot{m} c_p}\right) x} \tag{2}$$

Where T_s is the hot surface temperature, and h_{avg} is the average heat transfer coefficient at the fluid-metal interface. From Equation (2) above, one can observe that the ideal channel cooling length, L_{ch}, is proportional to the fluid entrance temperature, hot surface temperature, and the following parameters:

$$L_{ch} \propto \frac{c_p \dot{m}}{sh_{avg}} \tag{3}$$

$$\dot{m} \propto \frac{c_p L_{ch}}{sh_{avg}} \tag{4}$$

Where $\propto$ indicates proportionality. Once the parameters affecting the equilibrium length are determined, the relationship can be re-written for the mass flow rate by fixing the cooling channel length. A similar approach can be used for the constant surface heat flux case. It is important to note that the variables in Equations (3) and (4) are not independent and therefore it is not possible to determine an exact solution from the relations provided in Equations (1) and (2).

The pumping power considerations of this study depend only on the geometry of the channels; they are independent of the fluid temperature. For a given flow area, a different pumping power will be required if the channel cross-section is a circle, square, or triangle. The channel's location in the block does not make a large difference. The flow is assumed to be incompressible, uniform, fully developed, and fully turbulent. Operating conditions are assumed so that the working fluid does not undergo a phase change.

Chapter V. 2-D Conduction

This chapter provides information and analysis on an attempt to develop insight into the role that conduction plays in this problem by looking at several different configurations for a 2-D cross-section of a heat sink.

A. Problem Formulation

2-D conduction in a square with a cooling channel was examined to get an idea of the role that conduction would play in this problem. For this scenario, the problem was formulated with the boundary conditions shown in Figure 6(a).

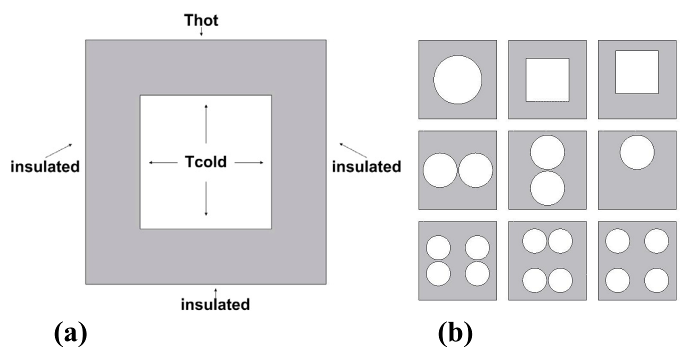


Figure 6. (a) Boundary conditions. (b) Shapes used for testing.

The top surface was held at a fixed hot temperature, the sides were modeled as adiabatic due to symmetry and the fluid-metal interface was modeled as a fixed cold temperature. No heat was transferred from the normal face of the metal block to the surroundings. Several different models were used to obtain insight into how channel shape and placement affect the heat transfer in the cell. The models used for this testing are shown in Figure 6(b). Each one has thirty percent of the volume available for channels for fluid to pass through with the exception of the one high circle. That case was chosen to determine how the heat sink would perform without the bottom circle.

B. Analysis

All the heat that enters the cell from the top is removed by the working fluid. Therefore, by assuming the change in temperature, ΔT, is equal to a change of one degree and that $Q_s = \dot{Q}_s$, the mass flow rate necessary to remove the heat was determined using the relationship:

$$\dot{m} = \frac{\dot{Q}_s}{c_p \Delta T} \tag{5}$$

From this necessary mass flow rate, as well as the flow area, the necessary pumping power was determined by dimensional analysis.

$$P = \frac{\dot{m}^3}{(\rho A_{flow})^2} \tag{6}$$

Here P is pumping power, V is velocity, p is the pressure, A_{flow} is the flow area, and ρ is the density of the fluid. To compare the different designs, the ratio of heat removed to pumping power was used.

C. Results

Using the boundary conditions given in Figure 6(a) with air as a working fluid, steel as the block metal, a hot temperature of 100°C, and a cold temperature of 25°C, computational fluid dynamics (CFD) software was used to run analyses on the models shown in Figure 6(b). After each simulation, the heat flux at the top surface was obtained from the CFD program. The pumping power was determined from Equation (6). The results are given below in Table 1.

Table 1. 2-D Conduction Results

	$\dot{Q}_s$ (W)	$\dot{Q}_s / P$ (dimensionless)
1 circle	147.89	0.58
1 square middle	155.32	0.61
1 square high	252.2	**0.99**
2 circle horizontal	127.07	0.06
2 circle vertical	309.75	0.15
1 circle vertical	309.45	n/a
4 circle horizontal	283.67	0.02
4 circle vertical	179.84	0.02
4 circle equal	295.15	0.02

D. Discussion of Results

As one can observe from Table 1, designs with one channel greatly outperformed their competitors. This is to be expected from the rough pumping power estimate produced by Equation (6). A channel size reduction of one half results in a pumping power value four times greater than the original. This result overpowers the increased heat removal rate that is obtained from designs with additional channels.

Designs with channels closer to the hot surface remove significantly more heat than those further away. This occurs because the driving force behind the heat flux is the temperature gradient as illustrated below in Equation (7).

$$\dot{Q} = kA_n \frac{dT}{dy} \tag{7}$$

Where k is the thermal conductivity, T is the temperature, and A_n is the area normal to the heat flow. Since the thermal conductivity and area normal to the heat flow will be the same for each of the trials and the temperature difference between the hot surface and the cold surface will be the same as well, the only variable left to impact the heat flux rate is the vertical distance between the two surfaces. Therefore designs with a smaller distance between the hot surface and flow channels will remove more heat than those located further away. By comparing the 2 circle vertical and 1 circle vertical results, one can observe that essentially all of the heat is removed by the top channel, the lower one in the 2 circle vertical case offers a negligible benefit.

Chapter VI. Performance Curves

This chapter provides information and analysis on an attempt to develop insight into the role that different parameters play in determining an optimal heat sink by generating performance curves.

A. Rationale

From the results of the 2-D conduction tests, it was determined that 3-D effects due to length must be considered. By considering the effects of fluid motion, the problem became one of conduction in a solid with forced convection through internal channels as opposed to pure conduction through a solid block. This work was done to gain insight into the relationship between input parameters such as the mass flow rate and channel layout, and output parameters such as the back bottom wall temperature and pumping power required.

B. Problem Formulation

For this simulation, the two models shown in Figure 7(b) were used to generate performance curves for several different mass flow rates with both water and air as a working fluid. Steel was maintained as the block metal.

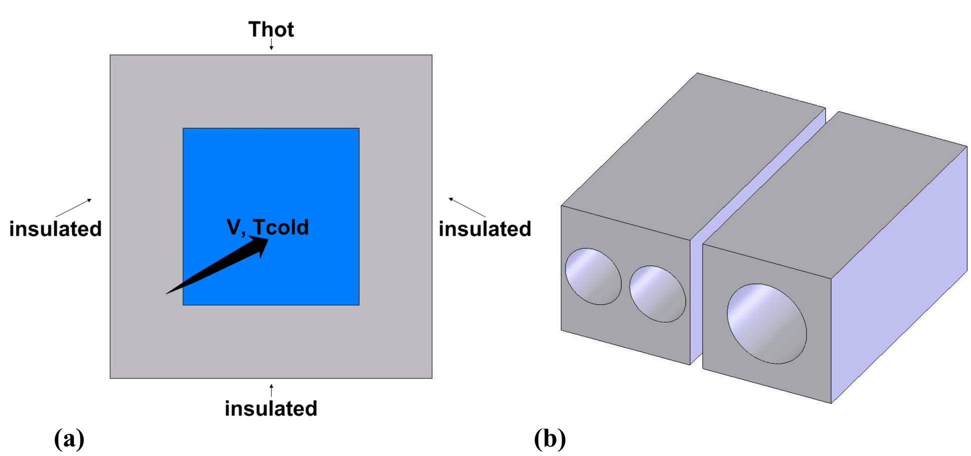
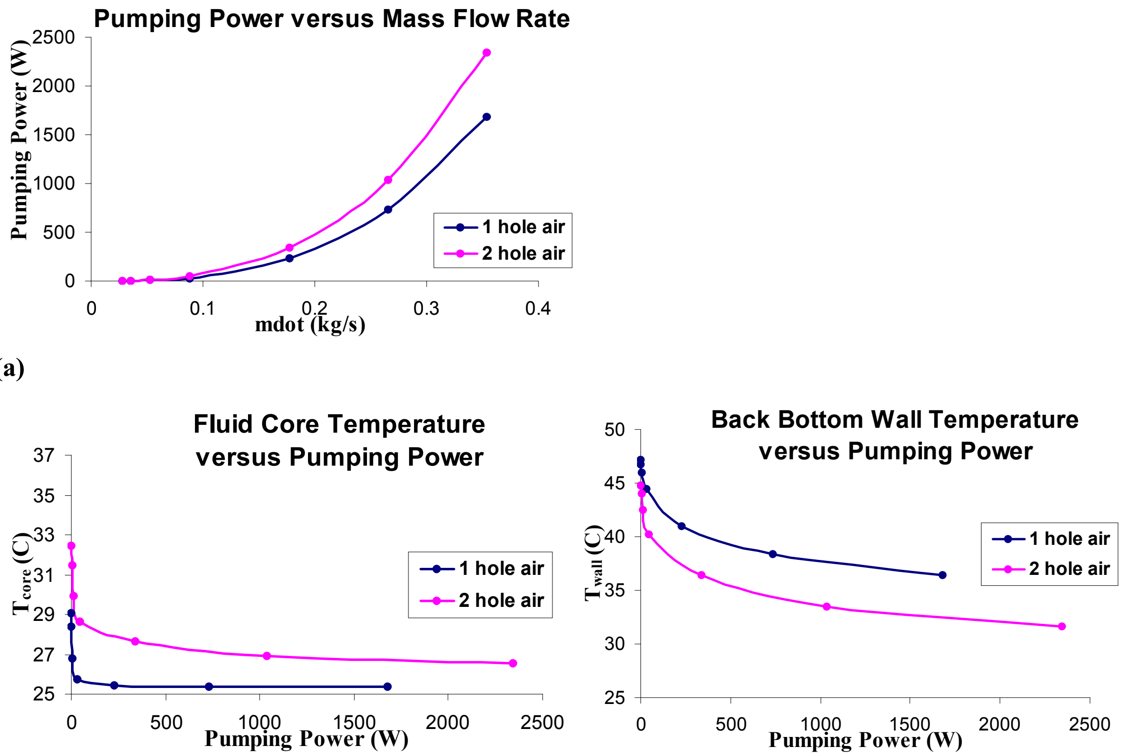


Figure 7. (a) Boundary conditions. (b) Models used.

As in the 2-D conduction trials, thirty percent of the block by volume was available for fluid to pass through. The boundary conditions shown in Figure 7(a) were used. These are essentially the same as the conditions used for the 2-D conduction trials with the exception of the fluid now having an inlet temperature and mass flow rate rather than a fixed temperature at the fluid-metal interface. The hot temperature was chosen as 50°C and 25°C was used for the cold fluid temperature.

C. Performance Curves

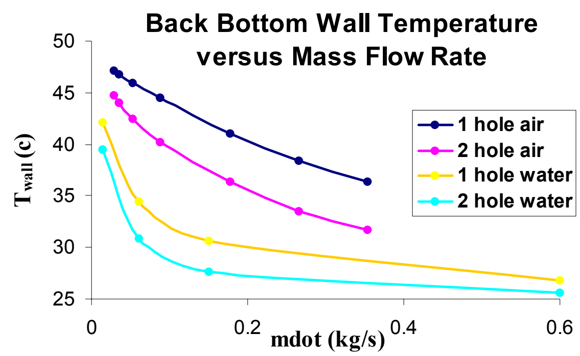
Figure 8 shows the different performance curves that were generated by these simulations.



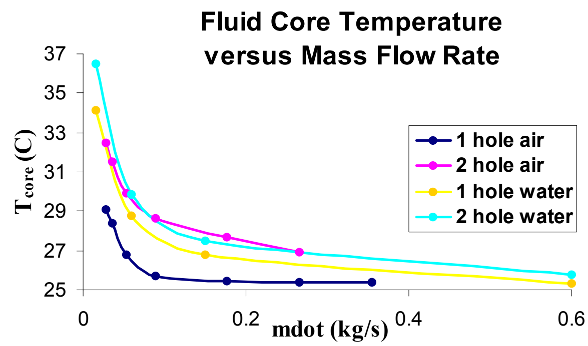
(a)

(b) (c)

Figure 8. Performance curves.



(d)



(e)

Figure 8 Cont'd. Performance curves.

D. Discussion of Curves

From the performance curves above one can observe several trends. Figure 8(a) shows that the pumping power rises in an exponential fashion with mass flow rate. As pumping power increases, the back bottom wall temperature and the fluid core temperature are reduced; this can be seen in Figure 8(b) and Figure 8(c). The same result is true for increasing mass flow rate (cf. Figure 8(d) and Figure 8(e)). A lower core temperature indicates that the fluid is leaving the channel before becoming fully hot and is therefore not being used as efficiently as possible.

All the curves show monotonically increasing or decreasing values which will eventually approach an asymptotic value and therefore further work with other shapes was not performed. From the initial trends shown, one can observe that the pumping power for a given mass flow rate increases with the number of channels. Another trend present is that the core fluid temperature increases with the number of channels. Lower back bottom wall temperatures are obtained from designs using a greater number of cooling channels.

Water has a much higher specific heat capacity than air and therefore removes a much larger amount of heat for a given mass flow rate. Additional difficulty with using air for these simulations occurs because the incompressibility assumption can easily be violated by flows in the desired cooling flow rate regime. As a result of this testing, water was chosen as the working fluid for further testing and air was abandoned.

Another option associated with performance curves is to conduct a transient analysis to look at designs which keep the back bottom wall temperature below a critical value for a given time. The initial back bottom wall temperature will be a function of the geometry of the channel, the mass flow rate, the flow area, the hot surface temperature, and the cold fluid temperature. As steady state conditions are approached, the back bottom wall temperature will approach the hot temperature for intensive thermal loads. The back bottom wall temperature versus time curve will be different for each condition and one could generate data for this situation to determine which design has the lowest temperature at a characteristic time. This analysis is suited for devices which undergo an intense but short duration thermal load.

Chapter VII. Performance Metric

This chapter provides information and analysis on an attempt to determine an objective function for use in comparing designs. An objective function is very valuable because it allows one to use optimization tools to systematically determine an optimal design.

A. Rationale

For optimization purposes, it is useful to have a single quantitative performance measure which can be minimized or maximized as an objective function subject to specific performance constraints. With such a tool, which will henceforth be referred to as a performance metric, it is relatively easy to determine the merits of different designs. This allows one to compare more traditional designs with constant cross-section square ducts to recent designs employing helical tubes like the ones seen in Figure 9.

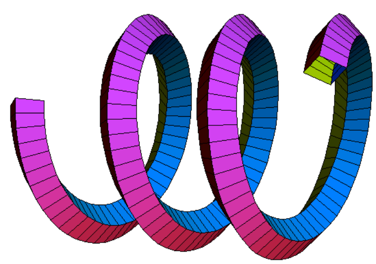


Figure 9. Helical tube with square cross section.
Palais, R. S., "Helix," *3D-XplorMath Space Curve Gallery*

Recent work studying fluid flow in helical tubes indicates that the higher heat transfer rate obtained from increased fluid mixing outweighs the increased pumping power penalty (Acharya, Sen and Chang 2001). Within these helical tubes several variables are able to be considered such as number of coils, number of axes coiled around, twist angle, and the coil radius. Investigating the viability of using coiled tubes in heat sinks is of great interest for further research.

B. Problem Formulation

The boundary conditions used in Chapter IV were kept for this analysis; they can be seen in Figure 7(a). Figure 10 shows the different models used for these simulations; note that they are 3-D but are shown in 2-D for ease of viewing.

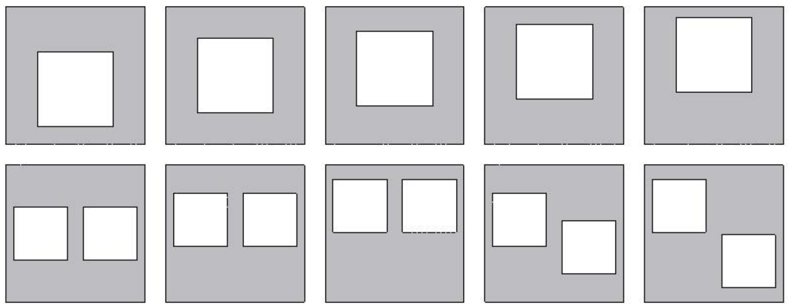


Figure 10. Geometries used for initial performance metric. The top row of designs are numbered 1,2,3,4,5 and the bottom row are numbered 6,7,8,9,10 respectively for analysis purposes.

Water was used as the working fluid and steel was used as the block metal.

C. Analysis

After observing that simply considering the metric of heat removed per unit pumping power did not prove adequate for optimization purposes, greater complexity was introduced in the performance metric. The following four goals were sought for an optimal heat sink:

i. Removes the greatest amount of heat
ii. The fluid just reaches a fully hot state as it exits
iii. The bottom wall temperature is as close to as possible while still below a critical temperature
iv. Uses the smallest amount of pumping power

Each of the above goals was written in the form of an equation and then non-dimensionalized by a relevant scale parameter. The beneficial aspects (goals i,ii,iii) were emphasized while the disadvantageous one (goal iv) was penalized. The following metric was obtained:

$$score = \frac{[\dot{Q}/\dot{Q}_0][T_{core}/T_{hot}][T_{bottom}/T_{critical}]}{[P/P_0]} \tag{8}$$

The scale parameters P_0 and $\dot{Q}_0$ were chosen as follows:

$$\dot{Q}_0 = \frac{k(T_{hot} - T_{in})LW}{H} \tag{9}$$

$$P_0 = \frac{\mu(\dot{m}/\rho A_{hole})^2 LA_{hole}}{H^2} \tag{10}$$

Where k represents the thermal conductivity of the metal, μ the dynamic viscosity of the fluid, L, W, and H are block dimensions. Q_0 was obtained by considering conduction through the block as a plane wall. A differential control volume was used to obtain the pressure difference over the channel length. The resulting equation for the pressure difference was used in Equation 6 to find the reference pumping power.

D. Results

Table 2. Performance Metric Results

Design number	$\dot{Q}$ (W)	T_{core}(C)	T_{bottom}(C)	p(Pa)	P(W)	Score (normalized)
1	1261.6	25.95	32.07	1.69	1.70E-4	0.727
2	1398.0	25.95	33.44	1.66	1.66E-4	0.824
3*	1645.2	25.98	33.32	2.22	2.23E-4	0.725
4	1568.5	26.22	34.73	1.70	1.71E-4	0.907
5	1695.8	26.28	35.25	1.67	1.68E-4	**1.000**
6	1469.5	26.83	25.59	3.34	3.34E-4	0.427
7	1777.1	27.47	30.36	3.40	3.41E-4	0.509
8	1980.5	27.64	30.90	3.41	3.42E-4	**0.566**
9†	1594.6	27.08	30.31	3.30	3.30E-4	0.470
10‡	1725.2	27.73	31.11	3.66	3.67E-4	0.461

* These values are believed to be caused by a numerical instability in the CFD program used and should not be considered valid.

†,‡ The variation in pressure value is indicative of poor CFD results; since the flow geometries are the same there should be less variability

E. Discussion of Results

The best designs were single channels located as close to the hot surface as possible. The additional pumping power requirements outweighed the increased heat flux benefit from more holes. There was not enough variation in the fluid core or back bottom wall temperatures between the different designs to make much of a difference in the final scores.

The fixed temperature boundary condition for the hot surface is not a good choice for the problem. The conduction of the heat through the metal to the top of the cooling channel is driven by the temperature gradient which is maximized by putting the cooling channel as close to the top as possible (cf. Equation (7) and Table 2). Such implicit dependence of the heat flux on the proximity of the cooling channel to the surface results in the lack of a baseline problem. Without such a baseline case it is not possible to conduct a rational optimization study. A better choice would be to use a constant heat flux boundary condition. A constant heat flux prevents designs with channels located close to the top surface from drawing upon an infinite reservoir of heat.

Designs which have a higher value of heat removed also have a higher back bottom wall temperature. This occurs because more heat is drawn in and not all of it is able to be removed by the top and sides of the cooling channel. Some heat is carried through the channel walls to the bottom section where it builds up in the area from the insulated bottom surface to the lower channel wall.

Another possible metric would emphasize designs which have the lowest average temperature possible. This would be done by re-writing the performance metric as

$$score = \frac{[\dot{Q}/\dot{Q}_0][T_{core}/T_{avg}]}{[P/P_0]} \tag{11}$$

Note that one could also use T_{max} instead of the average temperature. An advantage of this formulation is that it now seeks to minimize the average temperature of the heat sink which ensures the heat sink, and subsequently the device being protected by the heat sink, does not suffer failure due to thermal damage.

Chapter VIII. Skin Cooling of a Hypersonic Vehicle

This chapter contains a brief introduction to a thermal network for analyzing square channel heat sinks with a non-negligible wall thickness. Such an approach is well suited for heat sinks used to provide skin cooling of hypersonic vehicles. The description given is not complete but does contain important details for setting up the thermal network. It is hoped that a future student could complete and test this thermal network to determine whether it offers an improved solution to similar thermal networks.

A. Application Background Information

Objects flying at hypersonic speeds experience significant heating due to skin friction drag. Figure 11 shows the heat flux profile on the Hyper-X vehicle at Mach 7.

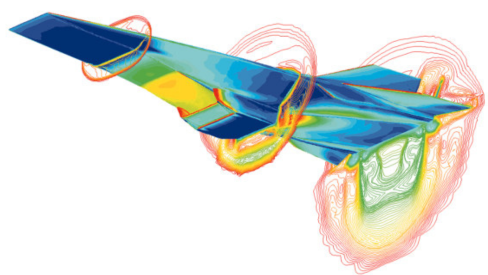


Figure 11. CFD simulation of the heat flux on the Hyper-X Vehicle at Mach 7.
NASA Dryden Flight Research Center. *X-43A Hyper-X Photo Collection.*
Blue represents low levels of heat flux and red represents regions of high heat flux.

From the heat flux profile shown in Figure 11, one can observe that the leading edges of the vehicle have very high heat flux values, as do locations where the flow turns. This poses a significant challenge for aircraft designers since the temperatures encountered are often above the melting point of materials available. This materials selection problem will only become more difficult as the maximum possible speed continues to increase. As an alternative to current high performance materials, heat sinks located just under the aircraft skin are being investigated. Research has shown that designs with an Aluminum-Niobium alloy and a working fluid with properties similar to water hold promise (Faghri 1995).

B. Problem Formulation

A unit cell approach is used with the boundary conditions given in Figure 12.

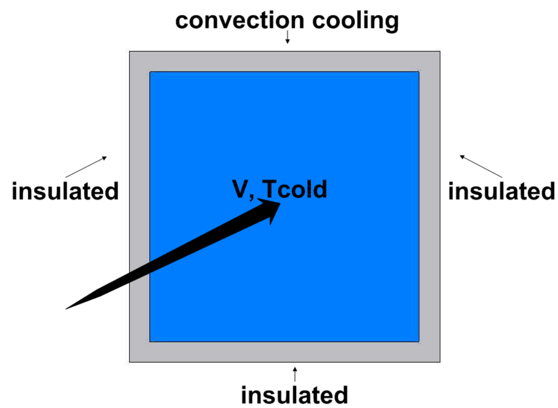


Figure 12. Boundary conditions.

Unlike the previously used fixed temperature boundary condition on the hot surface, the convection cooling boundary condition is more physically realistic. The shapes considered are limited to squares for this analysis. Symmetry is maintained and designs with n^2 channels are considered. This value can be extended from one until a sufficiently large n so that a critically small wall thickness value is reached. Several possible channel layouts are presented in Figure 13(a) to show the relative wall thickness differences.

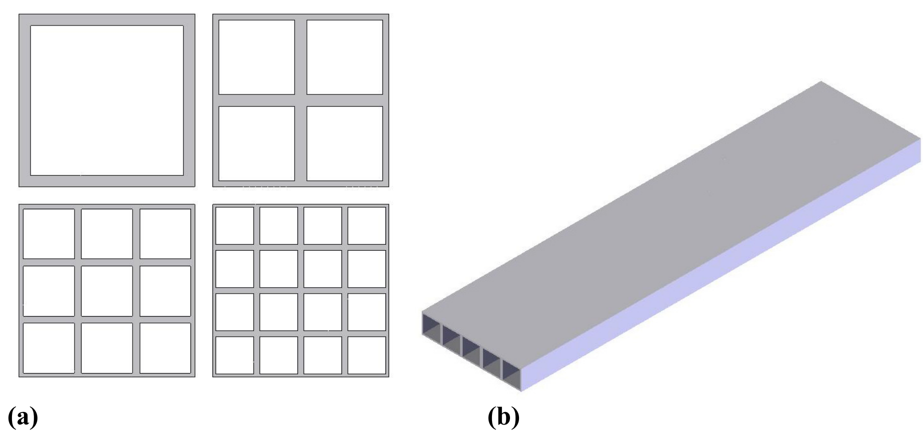


Figure 13. (a) Possible geometries. (b) Heat sink section.

The length will be much greater than the height for the cooling sections as shown in Figure 13 (b).

Valdevit et al. (2006) used a thermal resistance network analysis with a fin analogy to determine the temperature profile in a square channel and obtained good results. Their approach used separate 1-D analyses in the horizontal and vertical direction which is valid for designs with very thin walls but is less accurate in designs with thicker walls. The network being developed and described in this paper aims to further generalize the results by being able to obtain reasonable answers for designs where lateral conduction also plays a significant role and 2-D effects need to be considered. It is important to note that the approach being developed could in principle be applied to non-square channels but the analysis would be significantly more difficult because of the more difficult geometry.

C. Thermal Network Setup

Electrical resistance network analogies are a reliable tool for solving heat transfer problems. The information presented below represents initial work on developing a thermal network to model the heat transfer in thin walled square cooling channels. While the material presented is far from complete, key ideas are given that are hoped to lead to improvements over current models being used, such as the one developed by Valdevit et al (2006).

Figure 14(a) shows how a thermal network was set up to analyze a square geometry.

Table 3. Problem Parameters

Inputs
Material properties
Model dimensions
Fluid inlet flow rate
Fluid inlet temperature
Hot Temperature value
Outputs
Node Temperatures
Node Heat fluxes
Fluid exit temperature

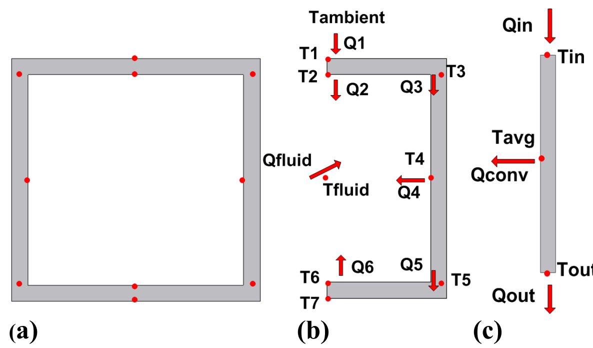


Figure 14. (a) Thermal resistance network nodes. (b) Simplified Thermal network with temperatures and heat fluxes shown. (c) Side element illustrated.

The nodes where heat fluxes and temperature values are calculated are shown as the red dots. By taking advantage of symmetry, the thermal network can be further simplified to the one shown with the relevant temperatures and heat fluxes in Figure 14(b). Figure 14(c) shows an expanded view of the side element, note that T_{in} corresponds to T_3 and T_{out} corresponds to T_5. This approach considers the temperature to vary only in the vertical direction; conduction in the horizontal direction is negligible. As these solid walls are very thin, this is not a bad assumption. To find T_{avg}, a differential control volume analysis was used to determine the temperature as a function of the vertical location in the element. The following equation is solved:

$$\frac{d^2T}{dy^2} + \frac{H_s}{W_s}(T_\infty - T) = 0 \qquad (12)$$

Where H_s is the height of the side piece, W_s is the width of the side piece, and T_∞ is the fluid temperature far from the wall. From this one is able to calculate the average temperature using the relationship

$$T_{avg} = \frac{1}{H_s}\int_0^{H_s} T(y)dy \qquad (13)$$

Where T(y) is the temperature variation in the vertical direction.

The top piece was modeled as shown in Figure 15(a). Rather than try to solve for the temperature distribution with such difficult boundary conditions, the problem was broken down using the principle of superposition into two simpler problems as shown in Figure 15(b) and Figure 15(c).

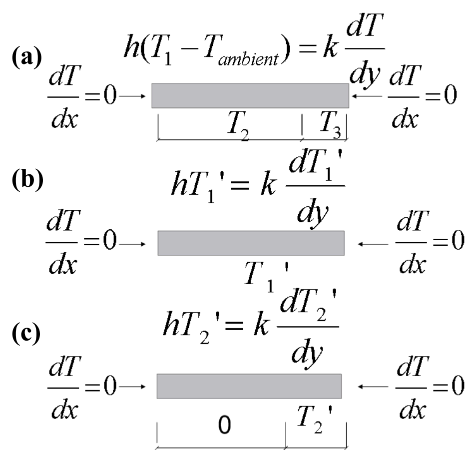


Figure 15. Heat conduction in the upper right corner.

(a) Case 1 boundary conditions.
(b) Case 2 boundary conditions.
(c) Case 3 boundary conditions.

Note that in Figure 15(b), T_1'=T_3-$T_{ambient}$ and in Figure 15(c) T_2'=T_2-T_3. A Fourier Cosine Series is used to represent the step function on the bottom surface boundary condition in case 3. Once one solves the governing equation for 2-D steady state heat conduction in a solid, Equation (14),

$$\nabla^2 T = 0 \tag{14}$$

for cases 2 and 3 throughout the rectangle to obtain the temperature distribution in the area, one can integrate the heat flux per length at the top and bottom surfaces to determine the total heat transferred across the boundary in each case. These values are then added together to form the solution to case 1. In this way the relevant heat fluxes are related to the node temperatures. A similar approach was used for the temperature distribution in the bottom piece, but the boundary conditions were slightly different. For that case the top boundary is T_6 until the side element part, T_5 in the side element part, and an insulated boundary condition is applied to the bottom surface. The boundary conditions for the left and right side remain insulated due to symmetry.

A steady-state heat balance is performed at the fluid-solid interface to relate the fluid temperature to the known values. The following relationship is obtained:

$$\frac{dT}{dz} = \frac{hs(T_{wall} - T)}{\rho c_p A_{hole} V} \tag{15}$$

Where h represents the convection coefficient, s is the perimeter of the channel, T is the fluid temperature at the given z position, and T_{wall} is the arithmetic average of the four channel side temperatures. The system of equations from the thermal network are solved to determine T(z) and then from that all other temperatures of interest are derived. For detailed information on how to set up and solve a thermal resistance network please see a heat transfer textbook. To view the detailed formulation of a similar application, consult the recent work done by Valdevit et al (2006).

Chapter IX. Conclusions and Future Work

This chapter presents the conclusions reached by this research and provides recommendations for future work.

A. Conclusions

Initial attempts at developing a systematic method to determine an optimal layout for a heat sink have proven unsuccessful. Both framing the problem and determining a valid performance metric for ranking designs have proven significantly more difficult than anticipated. Most of this work has been done with a constant surface temperature boundary condition which has consistently led to designs with one large square channel located close to the hot surface being shown as the best solution. This result is in contrast to current designs and leads one to believe that the problem to be solved has not been adequately formulated.

The greatest challenge remaining is how to frame the problem so that appropriate optimization techniques are able to be applied. As the skin cooling of a hypersonic vehicle application has shown, this research does have practical applications and will continue to become more important in the near future. While this research has not resulted in the desired performance metric, it is hoped that the lessons learned from this study may be used by future researchers to develop the potential of heat sinks made from cellular materials.

B. Future Work

There are many parameters that one is able to consider for a heat sink and it is believed that improvements to current conventional designs may be obtained. This paper has shown the difficulty in using a fixed temperature boundary condition for the hot surface; however simulations using a constant heat flux boundary condition have not been examined. Further research should be done in this area to determine what results may be obtained. Similarly, the introductory thermal resistance network approach presented above should be completed and simulations run to determine whether the results offer improvements to similar thermal resistance networks.

There are several technologies which have great potential to improve current heat sinks. Cellular materials allow one to manufacture designs down to a very small scale. Coiled tubes hold great promise in being able to offer greater performance than conventional straight tubes. Additional passive enhancements techniques, such as deliberately increasing the roughness of the tubes for greater fluid mixing could be considered. Active techniques could also be employed. Once a valid performance metric is established, the viability of more complex channel shapes such as triangles, rectangles, and ovals may be considered. Optimization techniques such as the Taguchi Method are expected to greatly reduce the number of simulations necessary for evaluating different designs.

Once the development of an optimal heat sink becomes more mature, one could move from focusing on this single component to integrating it into a full thermodynamic cycle for use in an application like skin cooling of a hypersonic vehicle. The feasibility of manufacturing several near optimal designs should also be investigated. If the construction of these designs proves viable, experimental testing to check the validity of the analytical model developed should be performed.

X. Bibliography

Acharya, N., & Sen, M., & Chang, Hsueh-Chia. (2000). Analysis of Heat Transfer Enhancements in Coiled-Tube Heat Exchangers. *International Journal of Heat and Mass Transfer*. 44, p. 3189-3199.

Afimiwala, K.A. & Mayne R.W. (1974). Evaluation of Optimization Techniques for Applications in Engineering Design. *AIAA Journal of Spacecraft and Rockets*. 11, 10, p. 673-674.

Afimiwala, K.A., Mayne, R. W., & Shah, R.K. (1978). *Heat Exchanger Optimization.* Proceedings of the Sixth International Heat Transfer Conference. Toronto, Canada. p. 185-191.

Blue Ridge Numerics. (2005). CFdesign examples guide version 8.

Blue Ridge Numerics. (2005). CFdesign technical reference version 8.

Blue Ridge Numerics. (2005). CFdesign user's guide version 8.

Cengel, Y. (2007). *Heat and Mass Transfer: A Practical Approach.* 3rd ed. NY: McGraw-Hill.

Faghri, A. (1995). *Heat Pipe Science and Technology*. NY: Taylor & Francis Group.

Fox, R.W., McDonald A.T., & Pritchard, P.J. (2004). *Introduction to Fluid Mechanics*. 6th ed. USA: Wiley.

Gibson, L.J., & Ashby, M.F. (1997). *Cellular Solids: Structure and properties*. 2nd ed. United Kingdom, Cambridge University Press.

Hedderich, C.P., & Kelleher, M.D., & Vanderplaats, G. N. (1981). *Design and Optimization of Air-Cooled Heat Exchangers*. ASME Transactions, Journal of Heat Transfer, Vol. 104. p. 683-690.

Incropera, F., & DeWitt, D. (1996). *Fundamentals of Heat and Mass Transfer.* 4th ed. USA: Wiley.

Kays, W. M., & London, A.L. (1998) *Compact Heat Exchanger*. 3rd ed. USA: Krieger.

Mangonon, P.L. (1999) *The Principles of Materials Selection for Engineering Design.*. Upper Saddle River, New Jersey: Prentice Hall.

NASA Dryden Flight Research Center. (1997). "Computational Fluid Dynamics (CFD) Image of Hyper-X research Vehicle at Mach 7 with engine operating." *X-43A Hyper-X Photo Collection*, Photo number ED97-43968-1. URL:

http://www.dfrc.nasa.gov/Gallery/Photo/X-43A/HTML/ED97-43968-1.html [cited 12 March 2007].

Palais, R.S. (2004-2006). "Helix," *3D-XplorMath Space Curve Gallery.* URL: http://rsp.math.brandeis.edu/3d-xplormath/SpaceCurves/helix/helix.html

Park, H., & Bowman, A. (2006). Governing equations used in coiled tube systems: [part I: Development of Standard/General forms]. *9th AIAA/ASME Joint Thermophysics and Heat Transfer Conference; San Francisco, CA; USA; 5-8 June 2006, AIAA Paper 2006-3617.*

Park, H., & Bowman, A. (2006). Governing equations used in coiled tube systems: [part II: Order-of-magnitude analysis of spiral coil systems]. *9th AIAA/ASME Joint Thermophysics and Heat Transfer Conference; San Francisco, CA; USA; 5-8 June 2006, AIAA Paper 2006-3616*.

Pluntze, S.C. & Eberhardt, D.S. (1998). *Application of Taguchi Techniques to the Evaluation of a CFD Solver.* URL: http://www.aa.washington.edu/faculty/eberhardt/Taguchi.pdf

Shah, R.K. (1978). *Compact Heat Exchanger Surface Selection Methods.* Proceedings of the Sixth International Heat Transfer Conference. Toronto, Canada. p. 193-199.

Shah, R.K. & London, A.L. (1978). *Laminar Flow Forced Convection in Ducts: a Source Book for Compact Heat Exchanger Analytical Data.* New York: Academic Press.

SolidWorks. (2005). Online tutorial. Student Edition Academic Year 2005-2006.

SolidWorks. (2005). SolidWorks 2005 Online User's Guide – Sp3.1. Student Edition Academic Year 2005-2006.

Valdevit, L., Vermaak, N., Hsu, K., Zok, F. W., and Evans, A. G. (2006) "Design of Actively Cooled Panels for Scramjets," *Proceedings of the 14th AIAA/AHI Space Planes and Hypersonic Systems and Technologies Conference; Canberra, Australia,* AIAA Paper, 2006-8069.

Webb, R.L. & Kim, N.H. (2005). *Principles of Enhanced Heat Transfer*. 2nd ed. NY: Taylor & Francis Group.

Appendix I. Biographical Sketch of Student

Matt de Stadler was born in Boston and raised in Newburyport, Massachusetts. He is the oldest of three children. Before UVA, Matt attended St. John's Preparatory School in Danvers, Massachusetts. It was there that he became interested in math and science, especially physics.

This interest drove him to major in aerospace engineering with a minor in applied mathematics. Since coming to UVA, Matt has been actively involved in several organizations, including the American Institute for Aeronautics and Astronautics, the American Society of Mechanical Engineers, Tau Beta Pi, Resident Staff, Math Club, Ballroom Dancing and several others. Also, Matt began research on this project with Professor Haj-Hariri in the 2nd semester of his third year.

Matt planned to attend graduate school from the time he arrived at UVA, and his coursework has further reinforced that goal. This thesis project is very valuable in that it prepared him well for his graduate studies. Next year he will attend the University of California San Diego to study computational and theoretical fluid mechanics.

UNDERGRADUATE THESIS PROJECT PROPOSAL
School of Engineering and Applied Science
University of Virginia

Geometrical Optimization of a Heat Exchanger

Submitted by

Matt de Stadler

Aerospace Engineering

STS 401

Section 3 (11 a.m.)

28 November 2006

Science, Technology, and Society Advisor: Ingrid H. Townsend

Technical Advisor: Hossein Haj-Hariri

On my honor as a University student, on this assignment I have neither given nor received unauthorized aid as defined by the Honor Guidelines for Papers in Science, Technology, and Society Courses.

Signed ____________________________

Approved ____________________________________ Date ________
Technical Advisor – Hossein Haj-Hariri

Approved ____________________________________ Date ________
Science, Technology and Society Advisor –
Ingrid H. Townsend

Abstract

This senior thesis project will examine different geometrical configurations to determine an optimal design for a heat exchanger. As heat exchangers tend to be very complex, the analysis will be simplified by considering a design which employs a repeating cell. Using the principle of superposition, the characteristics of the entire heat exchanger may be determined based on the characteristics of a representative cell. Within a cell, the following questions are posed: given a block of a material and a certain percentage of the volume available to put holes for fluid to pass through, where are the best places to put the holes? What shape should the holes be? This research is designed to answer those two questions. A purely theoretical approach will be employed, with testing and analysis performed using Computational Fluid Dynamics (CFD) software. The scope of this project does not include building and testing an actual model.

Energy efficiency is a very important topic now. With demand for power systems increasing annually, it is important that these systems be as efficient as possible. Heat exchangers are found in many areas, with applications ranging from satellites to nuclear power plants. Given such a wide range of uses, even a slight increase in efficiency will result in substantial savings. In addition to the cost savings expected, a more efficient heat exchanger will allow for systems to take advantage of improved performance capabilities.

Table of Contents

I. Rationale

This senior thesis project will examine different geometrical configurations to determine an optimal design for a heat exchanger. As heat exchangers tend to be very complex, the analysis will be simplified by considering a design which employs a repeating cell. Using the principle of superposition, the characteristics of the entire heat exchanger may be determined based on the characteristics of a representative cell. Within a cell, the following questions are posed: given a block of a material and a certain percentage of the volume available to put holes for fluid to pass through, where are the best places to put the holes? What shape should the holes be? This research is designed to answer those two questions. A purely theoretical approach will be employed, with testing and analysis performed using Computational Fluid Dynamics (CFD) software. The scope of this project does not include building and testing an actual model.

Energy efficiency is a very important topic now. With demand for power systems increasing annually, it is important that these systems be as efficient as possible. Heat exchangers are found in many areas, with applications ranging from satellites to nuclear power plants. Given such a wide range of uses, even a slight increase in efficiency will result in substantial savings. Hedderich, Kelleher and Vanderplaats (1981) state "Studies have shown that a poor choice of either the heat transfer surfaces or design parameters can more than double the costs chargeable to a heat exchanger (p. 683)." The possibility of this substantial penalty shows the care one must take in the design process. In addition to the expected cost savings, a more efficient heat exchanger will allow for systems to take advantage of improved performance capabilities.

A great potential use of a better heat exchanger is to reduce weight for space vehicles. Weight is incredibly important for launch vehicles. For example, Mangonon (1999) notes when discussing the space shuttle that it has been estimated that the savings per pound of weight in the orbiter vehicle structure or systems amounts to over $30,000 over the original predicted mission life of 10 years (p. 777). A more efficient heat exchanger will allow for the size and weight of the heat exchanger to be reduced. This is beneficial not only for the space shuttle but for many other applications.

II. Objectives

The ultimate goal of this project is to determine an optimal design for a heat exchanger. To do that, the following intermediate items must be completed:

- Determine performance metric to objectively rank different heat exchanger designs
- Investigate new designs for heat exchangers

III. Social and ethical implications

Heat exchangers are found in many varied applications making it difficult to say what results are likely to occur from technological advances. Since they are coupled with other devices, to fully understand their future impact, one must also understand the future impact of the systems they occur in. Areas of possible concern include practical, ethical and political results.

The main benefit of this research is that it will continue the development of more efficient heat exchangers. This will result in lower costs and allow for the improvement of many related systems. New performance capabilities are also likely to arise in related system components. This will have many benefits for the world community, beginning with less expensive energy production.

This technology will most likely benefit the middle class and the wealthy. While these improved designs will save money in the long run, they are going to have large capital costs which are likely to dissuade many potential users, especially those with little disposable income. It will take some time before the benefits of cheaper energy production reach the lower class. However, once these improved heat exchangers become standard, everyone will benefit greatly from the reduced cost of energy production. The military and power companies are likely to be the main beneficiaries as they have strong interest in the development of improved heat exchangers since these devices are integral members in many of their critical systems.

While the basic idea behind a heat exchanger is simple, many dependent variables affect the performance, making heat exchanger design somewhat of an art resulting in designs which tend to be very complex. Given that fact, the final product of this research is likely to be even more complex, which will lead to a further gap between lay users and the knowledge elite responsible for heat exchanger design. This widening knowledge gap is a trend to be aware of for the future.

It is hoped that manufacturing technology will continue to improve so that these more advanced designs can be built at standard factories. If not, new facilities will have to be developed to take advantage of this research. While this is likely to cost money, it will also create jobs to develop and maintain the new machinery.

Ethical difficulties arise with this project in that heat exchangers can be found in many items from air conditioners to nuclear power plants. Problems associated with a more efficient air conditioner will be much less worrisome than those with a more efficient nuclear power plant. With a cheaper heat exchanger, and hence an improved power cycle in general, less efficient energy sources could see wider use. One possible scenario is that the number of coal driven power plants could increase, contributing substantially to air pollution.

Heat exchangers are often members of a power cycle. These systems will improve as the components improve, and the improvement of the heat exchanger will lead to improvements in other components, for example, a better condenser. However it is hard to ensure that the devices which employ these power cycles will be operating in a benevolent manner. Currently, the future behavior this research will enable can not be accurately estimated. However, after considering the social and ethical implications, the expected benefits far outweigh the possible negative results and this trend is likely to continue into the future.

IV. Literature Review

A. Early History

Shah (1978) offers a substantial treatment of the history of heat exchanger development up to 1975. A brief summary will be given here. The study of heat transfer in laminar flow through a closed conduit was first made by Graetz in 1883, and later independently by Nusselt in 1910. Drew prepared a compilation of existing theoretical results for heat transfer data in 1931. Dryden et al. in 1932 compiled fully developed laminar flow solutions for ducts of various geometries. Later, Kays and London published a compilation in 1964 pertinent to compact heat exchangers. Porter compiled the laminar flow solutions for Newtonian and non-Newtonian liquids with constant and variable fluid properties in 1971 with the help of 30 British industries. The heat transfer literature up to 1967 was reviewed by Kays and Perkins who provided available results in terms of equations, tables, and graphs for design purposes (p. 2).

B. Technical Background

Heat exchangers have been in use for a very long time. A heat exchanger is a device which transfers thermal energy, heat, from a hot source to a colder one. Their ability to regulate temperature makes them suitable for a wide variety of applications, including nuclear power plants, satellites, reaction control for the pharmaceutical industry, air conditioners and electronics. This versatility makes heat exchangers a very valuable technology for the future. It also causes difficulty for design, as Afimiwala, Mayne and Shah (1978) state "The heat exchanger design problem is intricate because it involves many variables for geometry and operating conditions, also heat exchangers are designed for a variety of applications having different objectives (p. 185)."

This project draws heavily from three fields: fluid mechanics, heat transfer and optimization theory. Knowledge in each area will be important to ensure success as improvements in each of the three areas will help to develop a better heat exchanger. The choice of a material and working fluid are very important decisions as well, but for the purpose of this thesis, they will not be considered.

As Incropera and DeWitt (1996) state, heat exchanger research and development has had a long history which is still ongoing with efforts to improve design and performance. "With heightened concern for energy conservation, there has been a steady and substantial increase in activity. A focal point of this work has been *heat transfer enhancement*, which includes the search for special heat exchanger surfaces through which enhancement may be achieved (p. 618)." This thesis continues the spirit of Incropera's quote by aiming to determine improved geometries to increase performance. Webb and Kim (2005) discuss the benefits of enhanced heat transfer surfaces; they allow one to take advantage of one of the following options: size reduction, increased thermodynamic process efficiency which leads to lower operating costs, increased heat exchange rate for fixed fluid inlet temperatures, or reduced pumping power for fixed heat duty (p. 2). Enhancement techniques are separated into two areas, passive and active techniques. Passive techniques involve special surface geometries or fluid additives for enhancement while active techniques require external power such as electric or acoustic fields and surface vibration (Webb and Kim 2005). While many of these techniques have great theoretical promise, it is important to remember that a given enhancement geometry might not be able to be manufactured for all desired materials (Webb and Kim 2005). This thesis will focus on passive techniques.

Heat exchangers are typically classified according to their flow arrangement and type of construction (Incropera and DeWitt 1996). Many types of heat exchangers exist. The simplest of these is a concentric tube, which is essentially two concentric pipes with fluid flowing either in parallel or counterflow configuration. In counterflow the fluids flow in opposite directions, in parallel they flow in the same direction. Other possibilities include adding fins for increased surface area, or ensuring mixing occurs to obtain turbulent flow.

A more recent development has been the use of compact heat exchangers. These operate like any other heat exchanger but are characterized by a very large heat transfer surface area per unit volume. While many names exist for heat exchangers, the basic idea governing each one is the transfer of heat from a hot fluid, or surface, to a colder one. This can be accomplished in many ways, but this thesis will be limited to heat transfer from a hot gas to a cold liquid. Key factors in evaluating heat exchanger performance include the flow rates of the fluids, the temperature difference between the fluids, and the pressure difference between the inlet and exit for each flow.

In the past, difficulty in manufacturing confined most heat exchangers to using straight passageways with a circular or square cross-section for fluid flow. More recently there has been a massive undertaking to determine characteristics of non-circular ducts. Much of this data has been summarized by Kays and London (1964, 1978) and related literature. More recently there has been significant work done in the area of helical tubes (Acharya, Sen and Chang 2001). These tubes can have any cross section, most often a circular one is chosen, and twist to form coils around either one or two axes. The number of coils can vary from one, a very gentle curve, to an almost infinite amount of tangential curves like wires on a solenoid. The advantage of incorporating twist is that it increases fluid mixing which increases the heat transfer rate. The disadvantage of these designs is that they also increase the pumping power required. Particular care will be taken to model this twist as significant data is not easily available, in contrast to most conventional designs. Intuition leads one to believe there is an optimal way to employ these twisted tubes in a heat exchanger in order to get maximum mixing with minimum pressure increase. This is one of the key items to be investigated in this senior thesis project.

Optimization techniques are very important as well since there are so many variables and design options. Many possibilities exist for comparing the performance of different heat transfer surfaces, Shah (1978) lists several, categorizing them as "(i) comparison based on the Colburn heat transfer modulus and the apparent mean Fanning friction factor, (ii) comparison of heat transfer as a function of fluid pumping power, (iii) miscellaneous direct comparison methods, and (iv) performance comparisons with a reference surface (p. 196)." These techniques are good for single surface selection but do not result in an optimal heat exchanger. Another option is described by Afimiwala et al. (1978), one selects a single performance measure and defines it quantitatively seeking to minimize or maximize it, making it an 'objective function.' The customer then specifies performance constraints which results in the basic optimization problem being formulated as follows:

Minimize $f(\underline{x})$
Subject to $g_i(\underline{x}) \leq 0$ for $i = 1, 2, \ldots, p$
And $h_j(\underline{x}) = 0$ for $j = 1, 2, \ldots, q$

The functions may be either linear or non-linear. The vector $\underline{x}$ contains n unknowns which are to be adjusted to minimize or maximize the objective function f(x) (p. 185, 186).

A third optimization technique available is the Taguchi method. This method was developed in Japan to increase quality of manufactured goods and has since been adapted for many other uses. The Taguchi method is a statistical technique which determines the variables most critical to a given design. When used properly, it determines the importance of the interaction between two variables to the final answer. It does not tell how they interact but ultimately it allows the experimenter to save substantial time in identifying the parameters most relevant to the solution of the problem. Also, the technique allows one to tell which variables are unaffected by noise in the problem. The most practical advantage of the Taguchi method is that it will save substantial time, which is important as CFD simulations tend to be very time consuming.

Developments in cellular materials allow for consideration of significantly more complex heat exchanger designs. Cellular solids, a subcategory of cellular materials, are very relevant to this project. Gibson and Ashby (1997) define a cellular solid as “One made up of an interconnected network of solid struts or plates which form the edges and faces of cells (p. 2).” They offer a wide range of properties, making them applicable to such diverse applications as thermal insulation, packaging, structural support, and buoyant devices. Some advantages of cellular materials are a relatively low density, compared to the solid, a highly porous nature, and the ability to be tailored down to a very small scale. Also, cellular materials perform like “regular” materials on the macroscopic scale. The biggest advantage of cellular materials is that they allow one to construct a heat exchanger using the desired material, but with significantly more customizable geometries, especially on a very small scale (Gibson and Ashby 1997).

While much has been done with heat exchangers in the past, opportunities still exist for further improvement. The incorporation of appropriate twist has great promise for improved heat exchangers. With developments in cellular materials, the manufacturing capabilities are sure to increase substantially, which will allow an improved heat exchanger to be developed.

V. Research Plan

As this project was started in the spring of 2006, some of the main tasks have already been completed, including the following:

1. Define testing conditions
2. Investigate 2 dimensional problem to gain insight into relevant parameters
3. Build models for simulation which allow the collection of desired data
4. Develop grid-independent solution (within CFdesign)
5. Validate initial CFD solutions
6. Begin defining performance metric

With the above tasks already completed it is important that progress continue in the following areas:

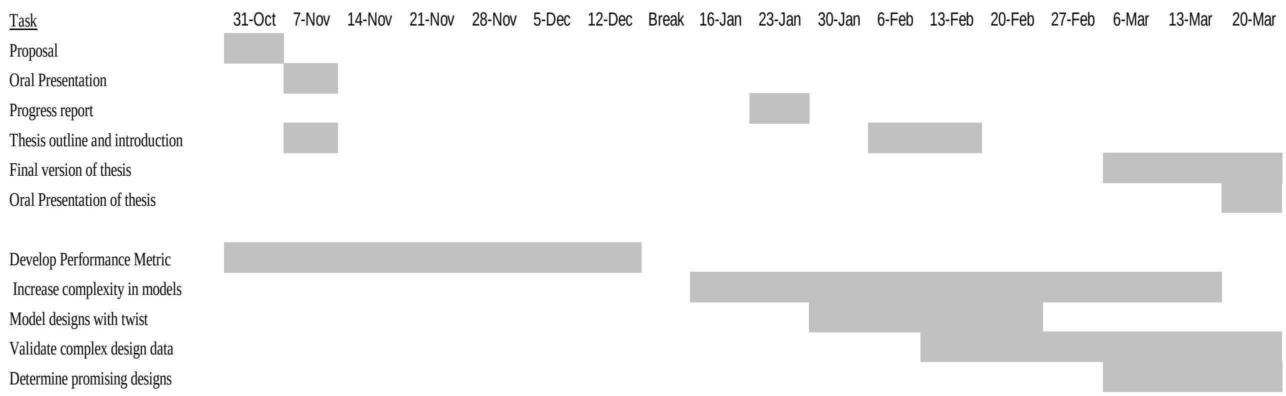
1. Refine performance metric
2. Continue to increase complexity in models
3. Model designs that incorporate twist
4. Ensure that more complex designs still produce valid results
5. Determine which designs show promise and should be considered further

Of the tasks remaining to be concluded, the most important is the performance metric. Without an objective measure to compare different designs it will be very difficult to determine an optimal solution. Once this step is complete, it will be relatively simple to produce data for use in comparing different designs.

VI. Schedule

Figure 1 below shows the schedule for this project.

Fig. 1 Gantt Chart



Expected Project Schedule

VII. Personnel

Technical Advisor: Hossein Haj-Hariri, Ph. D, Professor and Department Chair, Mechanical and Aerospace Engineering Department

To ensure completion of technical tasks, Professor Haj-Hariri and I will meet at least once a week to discuss my progress. Professor Haj-Hariri has over eighteen years experience as a professor and is a very well respected fluid dynamicist and applied mathematician. He conceived the idea of looking into twist as a method of increasing heat exchanger performance.

Faculty of the Mechanical and Aerospace Engineering Department

In addition to Professor Haj-Hariri's guidance, the faculty of the Mechanical and Aerospace Engineering Department will serve as additional technical resources to draw upon as necessary.

Society, Technology, and Society Advisor, Ingrid H. Townsend, Ph. D, Professor, Science, Technology, and Society Department

Professor Townsend will serve as an intelligent lay audience and assist with the social and ethical content of the project. Meetings will occur as necessary. She has over thirty years experience working with students on theses.

Joseph Wyatt, Graduate Student, Mechanical and Aerospace Engineering Department

Joseph is also working with Professor Haj-Hariri and his research uses similar tools such as the Taguchi Method, CFdesign and SolidWorks. It is expected that some discussion regarding optimization techniques and computer software used in this project will occur.

Student: Matthew B. de Stadler, Aerospace Engineering, Class of 2007

Matt is majoring in aerospace engineering with a minor in applied mathematics. He has taken substantial coursework in fluid mechanics and related subjects, including a graduate level computational fluid dynamics course. In addition to his strong fluid mechanics background, Matt also has a substantial background in applied mathematics with several courses taken in addition to the minor requirements. In the spring of 2006, he took an independent research course where he began work on this project. Also, he developed substantial programming skills as a result of his internships the past two summers at the Naval Research Laboratory.

VIII. Resources

As this project is computationally based, significant computer time is required. Two commercially available software packages, SolidWorks and CFdesign, will be used for this project. Both programs are available in the Mechanical and Aerospace Engineering Design Lab. SolidWorks allows one to build a physical model and then CFdesign uses that model to run CFD analyses. The MAE design lab is open twenty four hours a day, seven days a week, making it a very reliable resource. Relevant technical literature is available through the UVA library system.

IX. Expected Outcomes

At the completion of this project, it is expected that a performance metric will be developed that allows one to effectively rank different heat exchanger designs. This performance metric will be used to analyze several different geometries to determine which is most efficient.

After completion of this project, further research should be done in the following areas:

1. Increase complexity of model
2. Consider designs that do not use a repeating cell
3. Build a set of model heat exchangers and validate results
4. Look into the feasibility of building the optimal heat exchanger

5. Consider additional passive enhancement techniques

6. Consider addition of active enhancement techniques in addition to passive ones

Since the model to be analyzed includes a number of simplifications, a more general model could be developed. This will result in an even better heat exchanger than the one that is to be developed by this project. Also, one could build a set of models to experimentally validate the results obtained. When combined with more passive and active enhancement techniques, the performance of the resulting heat exchanger could increase dramatically. This possibility should be examined further, especially if one seeks to develop this heat exchanger for commercial applications.

X. **Bibliography**

Acharya, Narasimha, & Sen, Mihir, & Chang, Hsueh-Chia. (2000). Analysis of Heat Transfer Enhancements in Coiled-Tube Heat Exchangers. *International Journal of Heat and Mass Transfer*. 44, p. 3189-3199.

Afimiwala, K.A. & Mayne R.W. (1974). Evaluation of Optimization Techniques for Applications in Engineering Design. *AIAA Journal of Spacecraft and Rockets*. 11, 10, p. 673-674.

Afimiwala, K.A., Mayne, R. W., & Shah, R.K. (1978). *Heat Exchanger Optimization.* Proceedings of the Sixth International Heat Transfer Conference. Toronto, Canada. p. 185-191.

Blue Ridge Numerics. (2005). CFdesign examples guide version 8.

Blue Ridge Numerics. (2005). CFdesign technical reference version 8.

Blue Ridge Numerics. (2005). CFdesign user's guide version 8.

Fox, Robert W., Alan T. McDonald, & Pritchard, Philip J. (2004). *Introduction to Fluid Mechanics*. 6th ed. USA: Wiley.

Gibson, Lorna J., & Ashby, Michael F. (1997). *Cellular Solids: Structure and properties*. 2nd ed. United Kingdom, Cambridge University Press.

Hedderich, C. P., & Kelleher, M. D., & Vanderplaats, G. N. (1981). *Design and Optimization of Air-Cooled Heat Exchangers*. ASME Transactions, Journal of Heat Transfer, Vol. 104. p. 683-690.

Incropera, Frank, & DeWitt, David. (1996). *Fundamentals of Heat and Mass Transfer*. 4th ed. USA: Wiley.

Kays, W. M., & London, A.L. (1998) *Compact Heat Exchanger*. 3rd ed. USA: Krieger.

Mangonon, Pat L. (1999) *The Principles of Materials Selection for Engineering Design.*. Upper Saddle River, New Jersey: Prentice Hall.

Park, H., & Bowman, A. (2006). Governing equations used in coiled tube systems: [part I: Development of Standard/General forms]. *9th AIAA/ASME Joint Thermophysics and Heat Transfer Conference; San Francisco, CA; USA; 5-8 June 2006, AIAA Paper 2006-3617.*

Park, H., & Bowman, A. (2006). Governing equations used in coiled tube systems: [part II: Order-of-magnitude analysis of spiral coil systems]. *9th AIAA/ASME Joint*

Thermophysics and Heat Transfer Conference; San Francisco, CA; USA; 5-8 June 2006, AIAA Paper 2006-3616.

Pluntze, Stephen C. & Eberhardt, D. Scott. *Application of Taguchi Techniques to the Evaluation of a CFD Solver.*

Shah, R.K. (1978). *Compact Heat Exchanger Surface Selection Methods.* Proceedings of the Sixth International Heat Transfer Conference. Toronto, Canada. p. 193-199.

Shah, R.K. & London, A.L. (1978). *Laminar Flow Forced Convection in Ducts: a Source Book for Compact Heat Exchanger Analytical Data.* New York: Academic Press.

SolidWorks. (2005). Online tutorial. Student Edition Academic Year 2005-2006.

SolidWorks. (2005). SolidWorks 2005 Online User's Guide – Sp3.1. Student Edition Academic Year 2005-2006.

Webb, Ralph L. & Kim, Nae-Hyun. (2005). *Principles of Enhanced Heat Transfer*. 2nd ed. NY: Taylor & Francis Group.

Appendix I. Budget and Equipment

A Virginia Space Grant Consortium Undergraduate Aerospace Research Scholarship was awarded to help fund this research; the award totals $2,000 for materials and supplies for the 2006-2007 academic year. This money was used for a copy of SolidWorks, an external hard drive to store all results, and a laptop computer to be able to run simulations during the summer. A Ph.D. thesis was also purchased using these funds.

Appendix II. Biographical Sketch of Student

Matt de Stadler was born in Boston and raised in Newburyport, Massachusetts. He is the oldest of three children. Before UVA, Matt attended St. John's Preparatory School in Danvers, Massachusetts. It was there that he became interested in math and science, especially physics.

This interest drove him to major in aerospace engineering with a minor in applied mathematics. Since coming to UVA, Matt has been actively involved in several organizations, including the American Institute for Aeronautics and Astronautics, the American Society of Mechanical Engineers, Tau Beta Pi, Resident Staff, Math Club, Ballroom Dancing and several others. Also, Matt began research on this project with Professor Haj-Hariri in the 2nd semester of his third year.

Matt planned to attend graduate school from the time he arrived at UVA, and his coursework has further reinforced that goal. He intends to pursue a doctorate in mechanical engineering with a focus in fluid mechanics. This thesis project is very valuable in that it prepares him well for his graduate studies.

Appendix III. Outline of Thesis Report

Geometrical Optimization of a Heat Exchanger

I. Introduction/Rationale

This section will discuss why the current research is being done. Also, information from relevant technical literature will be discussed.

- I.1 Rationale
- I.2 Objectives
- I.3 Brief History
- 1.4 Literature Overview / Background material
    - i. Introduction
    - ii. Heat exchanger background
    - iii. Fluid mechanics background
    - iv. Optimization techniques
    - v. Cellular materials background
    - vi. Concluding paragraph

II. Social and Ethical Implications

This will talk about the related social and ethical concerns with this research.

- I.1 Part of a whole concept
- I.2 Expected Benefits
- I.3 Benefits the rich
- I.4 Will increase complexity
- I.5 Ethical nature depends on usage

III. Problem Statement

This section will contain details of how the problem to be solved was set up.

- I.1 Discuss framing of the problem
    - i. Include schematic of repeating cell
    - ii. Include schematic showing boundary conditions and
    - iii. Include variable list
- I.2 Discuss relevant assumptions

IV. Implementation

This section will contain details on how this problem was solved.

- I.1 Decision to examine results computationally
- I.2 Background information on SolidWorks and CFdesign
    - i. Discussion on some of the aspects of programs (i.e. lower predicted drag with CFD, difficulty of turbulence modeling)
- I.3 How results are obtained from simulations
- I.4 Verification of results

V. Results

This section will discuss the results achieved.

- I.1 Results obtained

VI. Conclusions

This section will commend on the significance of the results and make recommendations for further research.

I.1 Discussion of results
I.2 Significance of results
I.3 Overall conclusions
I.4 Recommendations for further study