

Thermal-Hydraulic Performance of Microstructured Heat Exchangers

Alex J. Heltzel

PC Krause and Associates, Inc.

Copyright © 2008 SAE International

ABSTRACT

Three-dimensional conjugate heat transfer models are built to predict the steady-state performance of microscale pin-fin and cross-flow heat exchangers with hydraulic diameters on the order of 100 μm . Modeling, meshing, and segmentation techniques are presented to allow for macroscale simulation of the microstructured devices. The effect of variation in geometric and flow parameters is investigated. Hydraulic and thermal predictions are compared to published experimental and extended beyond the limited range of test data to provide performance within a wide parametric range. A discussion of the dominating and relevant thermal transport mechanisms in both fluids and solid clarifies the routes to optimizing heat transfer in these small scale heat exchangers.

INTRODUCTION

The potential application of microchannel-based cooling devices to high waste energy density devices has attracted a great deal of attention since the work of Tuckerman and Pease brought it to attention [1]. Following this, the experimental work of Wu and Little revealed that conventional correlations fail to accurately describe heat transfer in microchannels [2]. The fundamental reason for the enhanced Nusselt numbers achievable in these devices stems from a thinning of the hydrodynamic and thermal boundary layers. Combined with the increased surface area available per unit volume, the macro-effect of microstructured cooling devices is a significant increase in volumetric heat transfer capacity. Niche technology areas requiring high-flux dissipation in localized areas stand to benefit greatly from the predicted performance advantages. Examples include high-brightness diode and laser diode array cooling, high-density integrated circuits, ultra-compact antennae, and directed energy weaponry, among others. The aerospace industry, which perpetually focuses on size and weight reduction, has also shown great interest in microstructure-based heat exchangers which have the potential to alleviate increasing thermal demands in strike aircraft and spacecraft.

Despite widespread interest for approximately two decades, only in recent years have advances in fabrication and imaging technologies contributed to applied engineering, fueled in part by a reduction in experimental uncertainty that led to earlier conflicting results. Recent reviews by Kandlikar [3], Cheng and Mewes [4], and Hidrovo et al. [5] give a comprehensive picture of the current state of micro-heat exchangers. Kandlikar and Upadhye [6] presented a parametric optimization study for cooling silicon chips with water, using a threshold temperature of 360 K. Several others have also investigated the effect of multiple design parameters on microchannel heat exchange performance [7-9]. Fundamental investigations into the continuity of single-phase flow have been performed both experimentally [10,11] and numerically/analytically [12,13]. The tradeoff between potentially steep pressure drop induction and exceptional cooling has also been a focus of investigation [14, 15]. This design parameter is often granted less attention, but is a critical factor for implementation of microstructured devices in aerospace applications, in particular. Quantized in the form of a friction factor, hydraulic loss is also shown to diverge from conventional correlations [16], adding additional urgency to a higher-fidelity method of prediction.

What the current investigation seeks to address is the noted deficit in applied engineering analysis using microstructured heat exchanger geometries that incorporate the cited knowledge for use in aerospace systems. The need for this type of analysis capability is immediate and multi-fold. Accurate and properly scaled performance can only be predicted using multi-dimensional, multiphase analysis tools provided by conjugate CFD/heat transfer modeling. The computational expense of such analysis precludes its use as a tool for quick design studies, even for independent heat exchanger functionality. In a system where the heat exchanger is but one of many critical components of the thermal circuit, its functionality must be predicted with minimal computing expense. Therefore, the need for accurate micro-heat exchanger scaling and characterization is obvious, with the presence of such modeling data being an enabling factor to the development of valuable technologies. This paper serves as an extension and completion of a previous

article written to address these issues [17]. The application of micro-heat exchangers modeled in this work is geared toward the thermal management of Mach-2 to Mach-4 class strike aircraft.

MODEL DEVELOPMENT

Micro-pin fin heat sink for electronics cooling

Consider the schematic of a micro-pin fin heat exchanger shown in Figure 1. The exchanger geometry is identical to that described in the experimental work of Kosar and Peles [18]. The pins are fabricated from a silicon wafer using deep reactive ion etching (DRIE), and bonded to a Pyrex cover. A detailed description of the fabrication and experimental procedures is found in the original paper [18]. The heat load is incident from bottom of the wafer, conducting up through the pin fins and convected to the cooling fluid. The pin fins have a diameter of $99.5\text{ }\mu\text{m}$, and lateral and axial pitches of 1.5. Pin length of $\sim 260\text{ }\mu\text{m}$ leaves $\sim 140\text{ }\mu\text{m}$ thick base in the $400\text{ }\mu\text{m}$ wafer. The length of the exchanger with machined pin fins is 1.0 cm, with an additional 4 mm inlet and exit plenum. The width of the pin fin channel is 1.8 mm, allowing for 12 pins per staggered row.

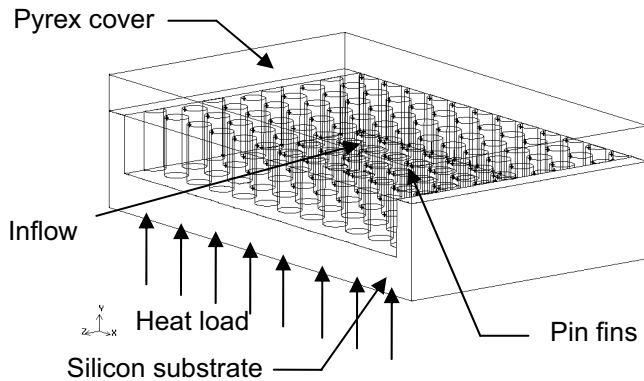


Figure 1. Schematic of micro-pin fin heat sink for electronics cooling

The geometry was solid modeled to these specifications with the exception of a shortened inlet and exit plenum. Due to the high aspect ratios and close tolerances in the experimental device, the model was segmented for a series simulation. This technique involves performing a CFD analysis on an inlet stage with plenum, recording the thermal and fluid dynamic profiles at the “exit”, and mapping them on to the “entrance” to a middle stage. An appropriate number of middle stage simulations are performed in series, mapping profiles from exit to entrance, after which a final exit stage simulation is performed, incorporating the exit plenum. It is critical that the stage boundaries cut through a pin row, providing exit profiles at the point of maximum velocity and minimum pressure. A stage boundary placed at the other point of velocity field symmetry- between pin rows- would neglect the presence of downstream pins and

lead to unrealistic pressure fluctuations and inaccurate predictions. Inlet and middle stage mesh segments are shown in Figure 2. Meshes were constructed using hexahedral/tetrahedral hybrid schemes in the commercial software Gambit, with each segment containing approximately 925,000 cells. Each mesh segment is 1.2 mm in length, requiring nine segments per simulation of the full-scale device. With completion of the individual stage simulations, the data is assembled in the visualization environment Fieldview for the acquisition of performance data of the device as a unit.

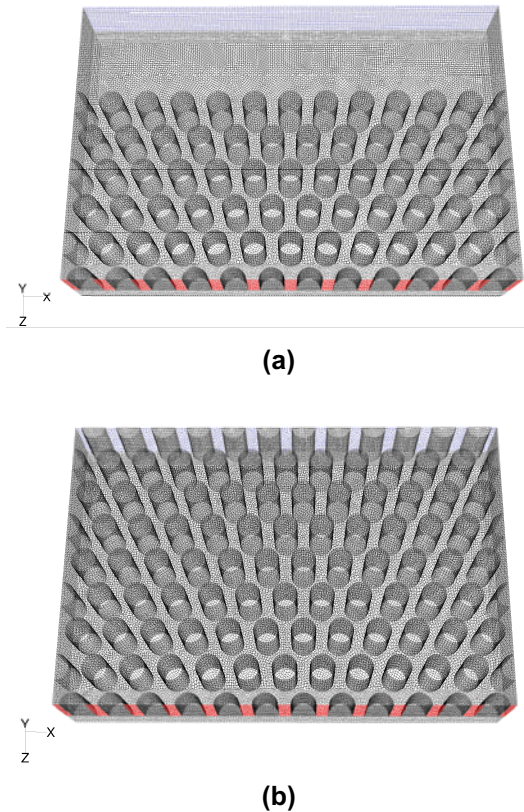


Figure 2. (a) Entry-stage mesh and (b) middle-stage mesh for micro-pin fin heat sink

Dual-phase microchannel heat exchanger

Whereas the micro pin-fin heat sink uses a single cooling fluid, the cross-flow heat exchanger designed and fabricated by Harris et al. [10] utilizes the perpendicular flow of gas through slots between to transfer heat to/from a liquid flowing perpendicularly through microchannels down the length of a panel. Note the distinction between the dual-phase device discussed here and the two-phase devices referenced above which utilize phase change heat transfer. A schematic of this device is presented in Figure 3. The heat exchanger is fabricated using a combination of LIGA and conventional machining processes. Heat exchangers have been constructed out of polymer (PMMA) and nickel [10], with plans to use ceramic, aluminum, or ceramic/metal

composites with higher thermal conductivities. Detailed fabrication and experimental procedures are found in the original papers [10, 19]. The solid model generated for this work simulates heat exchanger panels 3 mm thick, containing microchannels with cross-sections of 300 μm x 2.8 mm. Cross-flow channels of thickness 300 μm are modeled, with microchannel wall thickness of 50 μm . Total microchannel lengths of 10 cm are simulated in this work.

The CFD simulation discards any approximation for the gas cross flow such as a convection heat transfer coefficient, in favor of full CFD solutions for both the liquid phase in the microchannels and gas phase in the slots. Conduction through microchannel walls is also solved within the constraints of the energy equation (4). The importance of simulating the dynamics of both fluids becomes apparent when considering the application of the heat exchanger.

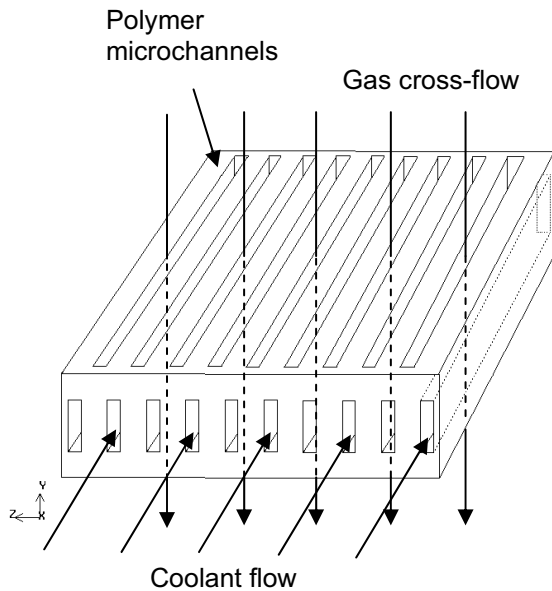


Figure 3. Schematic of micro-cross flow heat exchanger

In practice, multiple panels would be stacked, creating a user-defined number of layers for a given heat load. In the case of gas as the hot fluid, a significant amount of energy will be transferred to the liquid as the gas passes through each panel layer. The temperature profile of the air, and hence performance of the heat exchanger as a whole, will then be greatly dependent on the number of panels incorporated in the exchanger. This dependency cannot be accounted for accurately in a simple convective relationship where heat exchange is proportional to a fixed temperature gradient. However, CFD-resolved velocity and temperature profiles of hot gas exiting the slot of the first panel can be mapped onto the inlet slot of a second panel, and so on. In this way, physics-based scaling laws can be acquired that accurately describe the performance of cross flow micro-heat exchanger panels when assembled for realistic applications.

Figure 4 presents a unit computational cell out of which full scale cross-flow solutions are generated. Lateral symmetry in both geometry and flow field gradients allows the use of simply half of one gas slot, one microchannel wall, and half of one liquid microchannel (with microchannel walls at top and bottom of liquid channel). Slip viscous conditions and adiabatic thermal conditions at the symmetry planes dictate appropriate flow field and temperature field solutions for the entire channels. The device is segmented in the liquid flow direction similarly to the micro pin-fin heat sink. Each mesh segment is 20 mm in length, requiring 5 simulations in series for a panel 10 cm in length. The technique again involves mapping the liquid exit velocity and temperature profiles onto the inlets of the succeeding segments. Air inlet profiles are mapped from the exits of the panel "above" the panel currently being simulated, though constant conditions are used in the case of first panel simulation. Each 20 mm x 3 mm x 350 μm segment volume contains approximately 1.06 million hexahedral cells. Each assembled solution for a 10 cm panel then represents a 5.3 million cell simulation. It is important to note that this segmentation methodology is only appropriate for a device where the flow and temperature fields are constrained in such linear and symmetric fashion. A combustor could not be simulated accurately by segmenting the mesh, for example, where every point in the solution field is dependent on every other to a high degree.

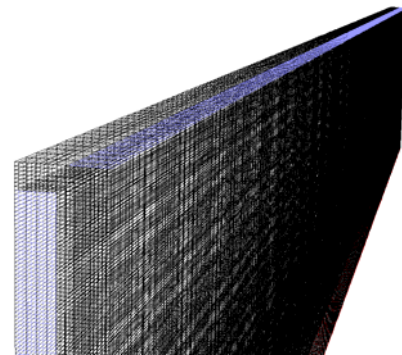


Figure 4. Computational mesh for micro-cross flow heat exchanger

The three-dimensional steady-state equations of continuity, momentum, and energy are satisfied by the commercial CFD software Fluent as follows:

$$\nabla(\rho \vec{V}) = 0 \quad (1)$$

$$\vec{V} \cdot \nabla(\rho \vec{V}) = -\nabla P + \nabla \cdot (\mu \nabla \vec{V}) \quad (2)$$

$$\vec{V} \cdot \nabla(\rho \cdot c_p T) = \nabla \cdot (k \nabla T) \quad (3)$$

where zero velocity in the solid domain reduces equation (3) to: