Simulation of 3-D Conjugate Heat Transfer in a rectangular cooling channel.

Ravinder Singh, Jani Pallis
Department of Mechanical Engineering
University of Bridgeport, Bridgeport, CT

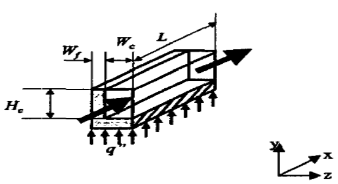
Abstract

Over the course of the past decade, a number of investigations have been conducted to better understand the fluid flow and heat transfer in channel heat sinks.

In this study full three-dimensional (3-D) conjugate heat transfer analysis is performed by computational fluid dynamics (CFD) codes for evaluating the thermal performance of rectangular cooling channel (heat sinks).

Introduction

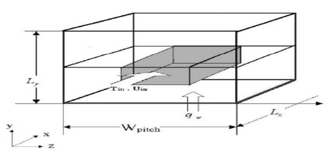
The present study conducts numerical study of heat transfer inside cooling channel with single level flow arrangements.



Rectangular channel heat sinks feature a high convective heat transfer coefficient, which is beneficial for various application where a very high rate of heat transfer is required.

In the current investigation, a detailed numerical simulation of the heat transfer occurring in Aluminum-based rectangular channel heat sinks has been conducted in order to optimize the geometric structure using a simplified, three-dimensional (3–D) conjugate heat transfer model [two-dimensional (2-D) fluid flow and 3-D heat transfer]. The investigated channel modeled with the length of 40mm and rectangular cross-section of 8mm X 6mm with the 6mm X 4mm coolant flow region.

Domain of the numerical simulation.



Mathematical Modeling

The following assumptions are made for the simplified model analysis:
●Force due to gravity is neglected. ●Heat dissipation caused by viscosity is neglected. ●Incompressible flow. ●Laminar flow (for simplified mathematical modeling). ●Negligible radiative heat transfer. ●Uniform convective heat transfer from the top surface (neglected for the mathematical model). ●Hydrodynamically fully developed but thermally developing flow.

For steady, fully-developed laminar flow in which,

$$\partial u/\partial x = 0,\ v = 0 \text{ and } w = 0.$$

The momentum equation for the liquid flow can be expressed as:

$$\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 x}{\partial z^2} = -\frac{1}{\mu}\frac{dP}{dx} = -\frac{1}{\mu}\frac{\Delta P}{Lx}$$

Where the boundary conditions are:

at the channel wall surface (no slip),	$u = 0$
at the inlet,	$x = 0,\ P = Pin$
at the outlet,	$x = 0,\ P = Pout$

The energy equation for the liquid is:

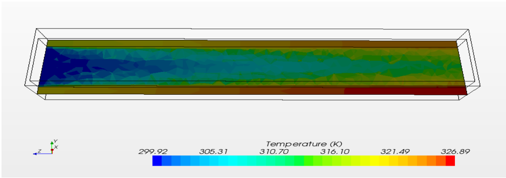
$$u\cdot \partial T/x = \frac{K}{Cp\,\rho}\left(\partial^2 T/\partial^2 x + \partial^2 T/\partial^2 y + \partial^2 T/\partial^2 z \right)$$

The heat conduction equation in the cooling channel:

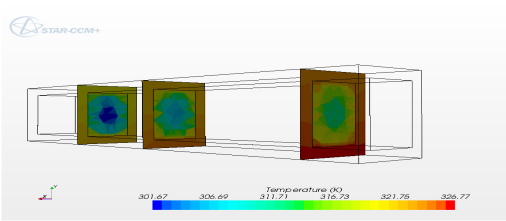
$$\partial/\partial x\,(K\ \partial T/\partial x) + \partial/\partial y\,(K\ \partial T/\partial y) + \partial/\partial z\,(K\ \partial T/\partial z) = 0$$

Simulation Results

Temperature Profile



Conjugate heat Transfer profile 3-D



Conclusion

The preceding analysis investigated the 3-D governing equations for fluid flow and heat transfer. The development work ongoing in the area of microchannels is broad, including different sizes and channel geometries, different liquids and gases, and microchannels with phase change. This analysis helped us in optimizing the channel geometry.