Simplified Modeling of Complex Compact Heat Exchanger Systems

Ben Zandi –TES International: (248) 362-2090 bzandi@tesint.com
Jeffrey Lewis –TES International
Hamish Lewis –TES International

Abstract

The goals of this paper are to develop and demonstrate an efficient modeling technique for the thermal analysis of compact heat exchangers for integration into more complex systems. The proposed simplified heat exchanger model can take advantage of an existing library of heat exchanger performance data, or alternatively, it can use existing heat exchanger performance curves. The ultimate utility of this methodology is to allow quick and accurate design decisions at system-level during package development by enabling the analyst to construct models and perform system-level thermal (and CFD) analyses in a matter of a few hours, rather than a few days (or weeks). The developed procedure has been demonstrated on a typical military aircraft electronics application, where the components are cooled by conduction of heat from the boards to "cold-plated" sidewalls. The proposed procedure is integrated into a thermal/CFD model of the above system.

Heat transfer between the sidewall and the working fluid is modeled using a fluid network. Each section in the network denotes a heat exchanger section and is linked to a corresponding section of the coldplate through using the "UA" of the heat exchanger.

Index Terms

Introduction

Electronic boxes with plug-in modules that are cooled by conduction of heat to air-cooled heat exchangers in the sidewalls are common in air transport equipments [1]. Typically, in these applications, a large amount of heat dissipated by the electronic components must flow through the PCB's (with many copper layers to enhance in-plane conduction) to the chassis sidewalls. The heat is then removed by the flow of conditioned air through the multiple-finned heat exchangers inside the sidewalls (called cold-plates). The component junction temperature is controlled by the effectiveness of the conductive heat path (i.e., component-to-board interface, conduction through PCB, conduction through clamps from PCB to chassis) and the efficiency of the heat exchangers. The main difficulty in modeling these problems lies in the modeling of exchangers. A numerical CFD model of the box that incorporates the details of the heat exchangers would be prohibitively large and impractical for most applications. In this study, we will use a fluid network heat exchanger model that is coupled with the main thermal / CFD solution scheme. The proposed simplified heat exchanger model can take advantage of an existing library of heat exchanger performance data, or alternatively, it can use existing heat exchanger performance curves. The ultimate utility of this methodology is to allow quick and accurate design decisions at system-level during package development by enabling the analyst to construct models and perform system-level thermal (and CFD) analyses in a matter of a few hours, rather than a few days (or weeks).

Problem Statement

The system was modeled as a sealed aluminum enclosure comprising of two cold plated sidewalls and 16 plug-in modules and one mother board. All boards were assumed to have two 8-oz copper layers to enhance conduction of heat to sidewalls. The total heat dissipation in the system was assumed to be 400 watts, applied on various components in the system. The simplified heat exchanger method was used to model the two sidewalls. Using this procedure the user need not model the fins in the heat exchangers and only needs to provide the mapping of the heat exchanger overall heat transfer coefficient (this may be available from test data or can be obtained by CFD modeling of the heat exchanger itself).

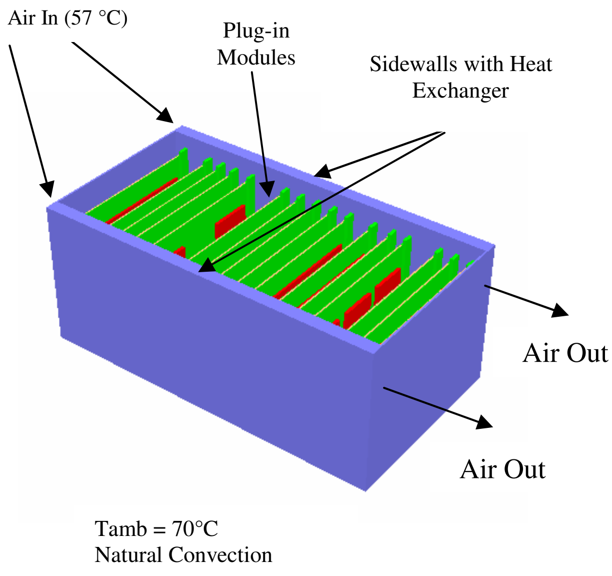

Figure 1: Electronics Box with Cold-Plate Sidewalls

Theory and Governing Equation:

The prediction of the heat transfer process requires understanding the values of the relevant variables (temperature, velocity, pressure, etc.) throughout the domain of interest.

Solid Regions:

In the solid regions, the energy equation reduces to the heat diffusion (conduction) equation [2]. This equation in its most general form is:

$$\rho C_p \frac{\partial T}{\partial t} = \left\{ \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) \right\} + q'''$$

Where:

x, y, z ≡ Spatial Coordinates
t ≡ Time
T ≡ Temperature
ρ ≡ Density
k ≡ Thermal Conductivity
q''' =Volumetric Heat Generation
C_p ≡ Specific Heat at Constant Pressure

Note that the volumetric heat generation term, q''', represents total heat dissipation in element. The solution for the above equation depends on the physical conditions existing at the boundaries of the system (boundary conditions) and on the system's conditions existing at some initial time (initial conditions).

Fluid Regions:

In the fluid region, it is necessary to solve a more general form of the energy equation to account for the convective effects.

$$\rho C_p\left(\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z}\right) = \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right) + q'''$$

Here, u, v and w represent the velocity components in x, y and z directions. The values of these components are not known and they must be computed by solving the continuity equation (conservation of mass) and the momentum equations (Navier-Stokes) equations.
For a Newtonian fluid with constant properties, these equations are:

(Continuity Equation)

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

(x-Momentum)

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial P_d}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)$$

(y-Momentum)

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial P_d}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$

(z-Momentum)

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial P_d}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) + \rho g \beta (T - T_a)$$

Where:

P_d ≡ Motion Pressure
μ ≡ Fluid Dynamic Viscosity
β ≡ Coefficient of Thermal Expansion
g ≡ Gravitational Acceleration (in z-Direction)

These equations must be solved simultaneously for temperature, pressure and three velocity components. The method used in this paper is based on "control volume" approach utilizing Patankar's SIMPLR (Semi-Implicit Method for Pressure Linked Equations – Revised) algorithm [3].

Heat Exchanger Boundary Condition:

Heat transfer between the sidewall and the working fluid is modeled using a fluid network. Each section in the network denotes a heat exchanger section and is linked to a corresponding section of the coldplate through using the "UA" mapping of the heat exchanger.

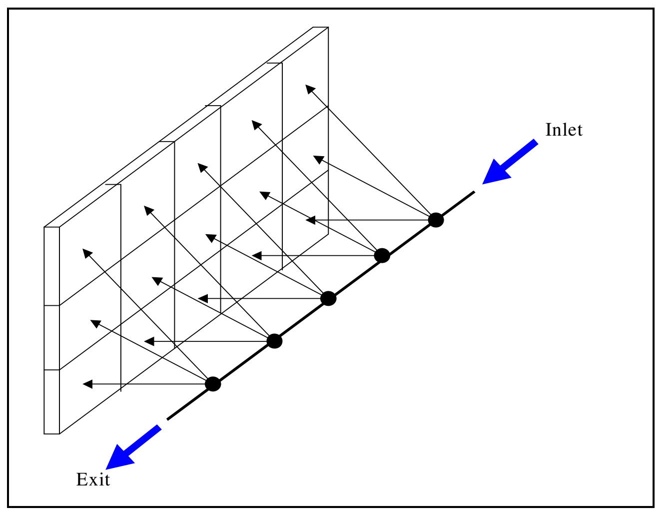

Figure 2: Simplified Heat Exchanger Network

$$q_{in} + \dot{m}.h_{in} = \dot{m}.h_{out}$$

$$q_{in} = \dot{m} C_p \left(T_{out} - T_{in}\right)$$

$$\Rightarrow \quad T_{out} = T_{in} + \frac{q_{in}}{\dot{m} C_p}$$

Similarly, for any two successive nodes:

$$T_i = T_{i-1} + \frac{q_{in}}{\dot{m} C_p}$$

Description of Analysis Software:

All modeling and numerical simulations for this study were performed using ElectroFlo® (EFlo), a commercially available software package developed by Zandi that has been used for over 10 years, although many features have been added more recently. The software used for this analysis is a finite volume package which incorporates the most important features for electronics cooling analysis. One of the key features is the use of coupled thermal/electrical algorithms in the solution. Thus the thermal problem is solved simultaneously with the electrical field. This is important because the resistance of an electrical circuit varies with temperature which then impacts the voltage and current fields in the circuit and the heat dissipated is a function of the current and resistance in each part of the circuit. The use of coupled thermal/electrical allows for local solution which results in a much more accurate point reading of temperature than would otherwise be possible. The solution method involves many iterations and the electrical properties and thermal properties are both re-calculated during each iteration. This coupling becomes of greater value with the increasing demands on the electronics used in many areas. The increased accuracy and solution speed afforded become invaluable.

The package has computational fluid dynamics capability, although it can be run either with or without this feature turned on. A patented radiation solver is also incorporated.( ) In many cases radiation can be a more significant contributor to heat transfer than natural convection. The blocking technique used allows a much more efficient solution than would other wise be possible.

In this study, we will use a fluid network heat exchanger model that is coupled with the main thermal / CFD solution scheme. The proposed simplified heat exchanger model can take advantage of an existing library of heat exchanger performance data, or alternatively, it can use existing heat exchanger performance curves. The ultimate utility of this methodology is to allow quick and accurate design decisions at system-level during package development by enabling the analyst to construct models and perform system-level thermal (and CFD) analyses in a matter of a few hours, rather than a few days (or weeks).

The analysis was run on a standard windows based PC.

Results:

Figure 3 shows three-dimensional temperature distribution results for the system. Figure 4 provides a temperature fringe plot for one cold-plated side wall. From this figure it can be seen that cooling air enters the heat exchanger at 57 °C and leaves the cold-plate at 68 °C.

Figure 4 presents temperature and flow distribution on a plane over the board number 4.

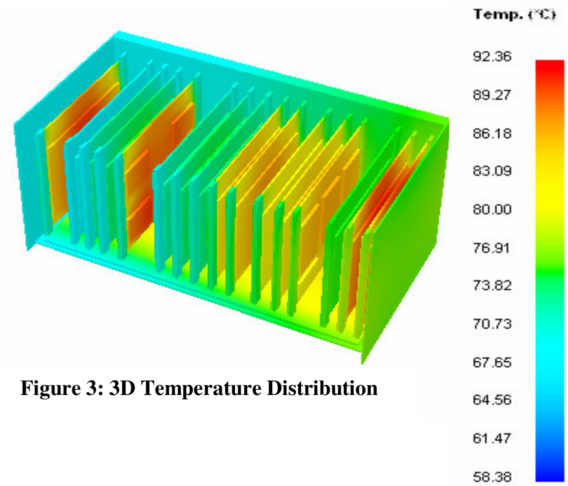

Figure 3: 3D Temperature Distribution

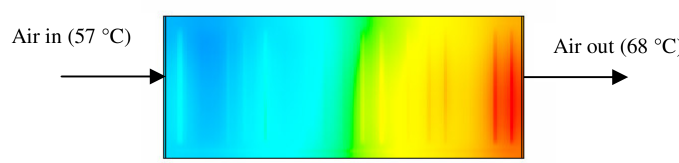

Figure 4: Sidewall Temperature

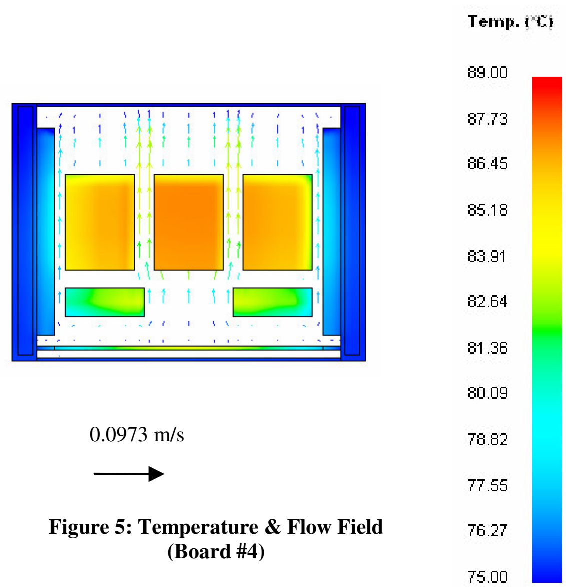

Figure 5: Temperature & Flow Field (Board #4)

Conclusions:

In this paper, a simplified approach for incorporating compact heat exchanger models into complex thermal/ CFD system models using a coupled fluid network methodology is developed and demonstrated. The simplified heat exchanger model takes advantage of an existing library of heat exchanger performance data, or alternatively, it can use existing heat exchanger performance curves. The ultimate utility of this methodology is demonstrated on a typical military aircraft electronics application, where the components are cooled by conduction of heat from the boards to "cold-plated" sidewalls. The proposed procedure is integrated into a thermal/CFD model of the above system.

References:

1. Steinberg, D. S., Cooling Techniques for Electronic Equipment, John Wiley and Sons, 2nd Edition, 1991.
2. Incropera, F. B., Dewitt, D. P., Fundamentals of Heat and Mass Transfer, John Wiley and Sons, 1985.
3. Patankar, S. V., "A Calculation Procedure for Two-Dimensional Elliptic Situations", Numerical Heat Transfer, Vol. 4, pp. 409-425, 1981.