

Application of Solution Mapping to Reduce Computational Time in Actively Cooled Power Electronics

Kirk T. Lowe^{*1,2} and Rao V. Arimilli²

¹Oak Ridge National Laboratory

²Department of Mechanical, Aerospace, and Biomedical Engineering, University of Tennessee

*Corresponding author: 414 Dougherty Engineering Bldg, 1512 Middle Dr., Knoxville, TN 37996, klowe1@utk.edu

Abstract: As power electronic applications advance, controlling operating temperature becomes more challenging. Some applications arise where the available coolant temperature is close to the maximum operating temperature of the power electronics. In these cases, new thermal management schemes must be considered. One such idea is to incorporate the heat sink directly into the packaging of power electronic modules.

COMSOL is used to predict the 3-D fluid behavior and 3-D temperature distribution within an actively cooled power electronic structure. A solution mapping method is implemented to more efficiently model axisymmetric flows that lie within the heat exchanger. The results show that the proposed design maintains the chip below the maximum operating temperature, but the fluid temperature exceeds its boiling point. Increasing fluid velocity will reduce the maximum fluid temperature below the boiling point. This design concept demonstrates the feasibility using high temperature coolant with power electronics.

Keywords: Extrusion Coupling Variables, Power Electronics Cooling, Solution Mapping, Convection Heat Transfer

1. Introduction

Cooling of electronic devices is crucial to their performance and reliability. In traditional packaging of power electronic devices, many layers exist to address the electrical requirements of the device. However, incorporating these packages into new applications can prove to be challenging, where the size and operation of the heat sink must be optimized. This task becomes even more important in applications where the ultimate coolant temperature is close to the temperature limit of the device. The excess thermal resistance must be removed.

In typical packages, as shown in Figure 1, the power electronic chip is soldered to a direct bonded copper (DBC) substrate. This substrate provides electrical insulation from the rest of the package. The DBC has three layers. The center is a thin layer of ceramic for electrical insulation. On either side of the ceramic, a layer of copper is bonded to provide solder surfaces for the chips and to connect the DBC to the package base. The DBC is soldered to a base plate to spread the heat out over a larger area and provide a more rigid structure to mechanically attach the module to traditional heat sinks.

The module is typically encapsulated in a plastic housing, and the chip is covered with a silicone gel to protect the wire bonds (not shown) from vibration and arcing. A thermal interface material (TIM) is placed between the power electronic module and heat sink to fill voids and promote better heat conduction to the heat sink.

By eliminating some of these layers, particularly the TIM, heat transfer can be greatly increased by reducing the thermal resistance. Moreover, in cases where the coolant temperature and the allowable operating temperature of the chip are close, new packaging topologies must be considered to ensure proper electrical operation and thermal control. One such design proposal is to incorporate the heat sink into the DBC layer as shown in Figure 2 [1].

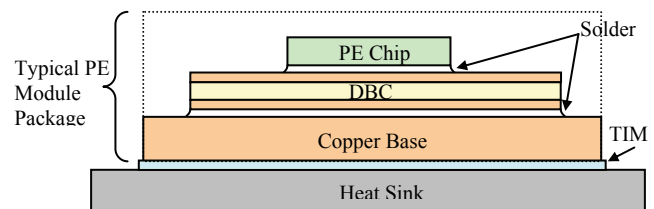


Figure 1. Typical Power Electronic Package

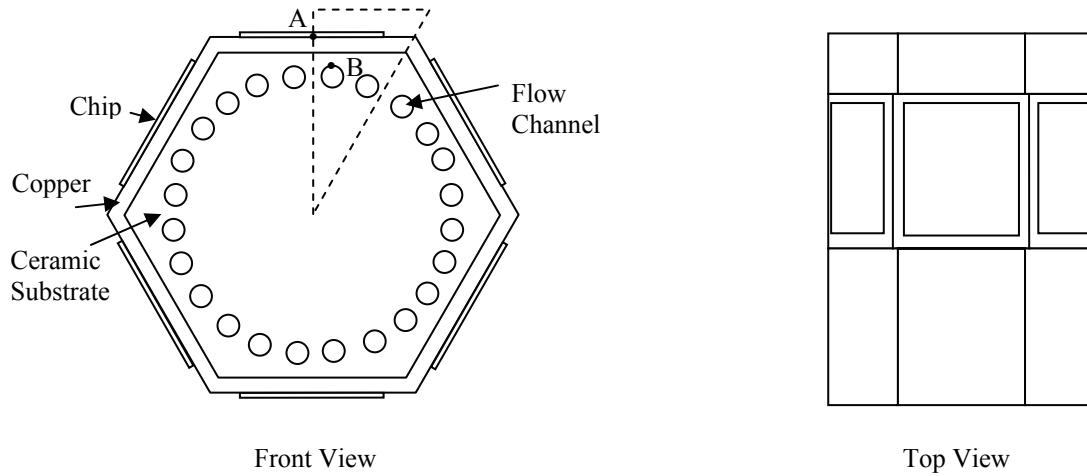


Figure 2. Schematic of Imbedded Heat Exchanger

The inner hexagon in the front view is the ceramic substrate. A copper layer surrounds the substrate but does not cover the full length of the hexagon prism and is not symmetrically placed in the axial direction. The chips are then soldered onto the copper layer. Coolant would flow through the circular channels that lie beneath the bonded copper surface.

The dashed line depicts a thermal symmetry section, representing a unit cell that is modeled. The objective of the study is to predict maximum chip and fluid temperatures at steady state operation. These temperatures must be kept within operational limits of the power electronic chip and the boiling point of the fluid, 150°C and 130°C, respectively. For lower loads and during transient power fluctuations, the resultant design will perform at even lower chip and fluid temperatures because steady state conditions correspond to the maximum operational load.

2. Model Setup

In general, the modeling would consist of a coupling of 3-D Incompressible Navier-Stokes and 3-D General Heat Transfer modules. It is possible to fully couple these models in COMSOL with temperature dependent physical properties and a buoyancy source term. In this case, the flow is forced convection so the buoyancy source can be neglected, and the physical properties are chosen to be constant. Therefore, full coupling of the flow and thermal models is not required. Instead, the velocity field

will be solved first and the resulting solution is used to solve the steady state temperature distribution. Even with this approach, the three dimensional nature of the design requires either a large multiprocessor machine, long iterative solution schemes, or a novel alternative to solve for the velocity and temperature fields.

The velocity field in each flow channel of the design should be identical given a uniform pressure at the inlet. Furthermore, the flow field in the channel is axisymmetric. Thus, a novel solution scheme is used to map a 2-D axisymmetric flow field solution onto the 3-D unit cell. The mapped flow field solution is then used to solve for the temperature distribution in the fluid.

2.1 Governing Equations

For the 2-D axisymmetric flow field, the governing equations are the incompressible Navier-Stokes and the continuity equations:

$$\rho \bar{u} \cdot \nabla \bar{u} = \nabla \cdot \left[-p \mathbf{I} + \eta (\nabla \bar{u} + (\nabla \bar{u})^T) \right]$$

$$\nabla \cdot \bar{u} = 0$$

where ρ is the density, \bar{u} is the velocity field, p is the pressure distribution, η is the viscosity and \mathbf{I} is the identity matrix. The fluid properties are chosen as constant, which are specified at a mean temperature approximated by an energy balance on the flow channel with a uniform wall flux.

Table 1. Thermo-physical Properties of Materials and Working Fluid

Material	Thermal Conductivity <i>W/m-K</i>	Density <i>kg/m³</i>	Specific Heat <i>J/kg-K</i>	Viscosity <i>Pa-s</i>
Ceramic 1	25	3700	800	-
Ceramic 2	160	3300	740	-
Ceramic 3	146	2850	1046	-
Ceramic 4	130	3100	720	-
Working Fluid	0.4	1006	3750	0.0006

These properties were considered sufficient because the thermal properties of the working fluid vary less than two percent over the maximum operating temperature range. These properties are listed in Table 1.

After the velocity field is solved, it is mapped to a 3-D cylinder, and the temperature distribution can be found. The steady state temperature distribution in the ceramic, copper, and chip is given by the heat equation with constant properties.

$$k\nabla^2 T + Q''' = 0$$

Where k is the thermal conductivity, T is the temperature, and Q''' is volumetric heat generation. Only the chip will have a volumetric heat generation applied. The ceramic material properties are specified in Table 1. For the ceramics, the properties are specified at the maximum allowable operating temperature of the chip. These values should yield conservative results because thermal conductivity increases with decreasing temperature. This model will also be evaluated for several different material candidates because of material availability, manufacturability, compatibility, and cost.

In the General Heat Transfer model, the coupling of the flow field and the temperature distribution is done through enabling convective heat transfer. The fluid in the flow channel experiences convection, which is governed by the energy equation:

$$\rho C_p \bar{u} \cdot \bar{\nabla} T = Q''' + k\nabla^2 T$$

where C_p is the specific heat of the working fluid and the velocity field, \bar{u} , is from the mapped flow field solution.

2.2 Extrusion Coupling/Solution Mapping

As alluded to earlier, the model requires the 3-D flow field to be resolved before solving the temperature distribution. Early models using iterative methods proved to be extremely time consuming and resource intensive. Models for the 3-D flow field alone required fine mesh refinement for adequate solution convergence. For just one of the flow channels, the model created near one million degrees of freedom, which was the upper limit of the computing resources available.

To speed up and in some cases enable, the flow field solution process, Extrusion Coupling Variables, a feature available in COMSOL, were used. A two-dimensional axisymmetric solution of the flow field was mapped onto three-dimensional cylinders, which are a part of the unit cell model as shown in the dashed triangle in Figure 1. The flow field in the cylinders was then resolved into u , v , and w components and used in the General Heat Transfer convection model. This process allows for a coarser mesh to be used to find the temperature distribution. The reduced number of degrees of freedom allows for more solver options, including direct solvers, to be used to find the temperature distribution. With the computing resources available, this process reduced the entire computational time by more than two orders of magnitude as compared to just solving the entire 3-D velocity field.

The first step in the solution mapping was to find the 2-D axisymmetric solution of the velocity field. For this solution, the inlet velocity was initially specified for a very low Reynolds number and was increased parametrically to progressively higher velocities by using previous solutions as the initial guesses. In these cases, the relative tolerance was 1e-8 for the UMFPACK direct solver.

This solution was then mapped onto the two cylinders in the unit cell model using Extrusion Coupling Variables. The general transformation from the 2-D to the 3-D took the u and v velocity components, which were in the r and z direction, respectively, and mapped r to the x - z plane and z onto the y -axis. Because the centers of the cylinders are not at the origin in the 3-D model, a coordinate transformation is used to shift the origin such that:

$$\sqrt{(x - x_i)^2 + (z - z_i)^2} = r$$

$$y = z$$

where (x_i, z_i) is at the center of the channel inlet. This process completes the solution mapping procedure. More detailed instructions are included in COMSOL - Help [2].

For convection to be implemented, the velocity field which is still defined in \mathbf{u} and \mathbf{v} components must be broken into \mathbf{u}' , \mathbf{v}' , and \mathbf{w}' components. The \mathbf{v}' component is still in the form of the original solution and requires no additional manipulation. The \mathbf{u} component, which was originally in the radial direction and was then mapped to the x - z plane, must be broken down into its respective \mathbf{u}' and \mathbf{w}' components.

The components, \mathbf{u}' and \mathbf{w}' , can be found by trigonometry as shown below. The angle must be found by shifting the coordinates in the channels back to the original origin. Subdomain variables x' and z' are defined, where the respective channel center is defined at (x_i, z_i) for each channel in the model.

$$\begin{aligned} x' &= x - x_i \\ z' &= z - z_i \end{aligned} \quad \theta = \tan^{-1}\left(\frac{z'}{x'}\right) \quad \begin{aligned} u' &= u \cos(\theta) \\ w' &= u \sin(\theta) \end{aligned}$$

2.3 Boundary and Mesh Parameters

The boundary conditions for the 2-D axisymmetric flow were alluded to earlier. In summary, the inlet has a specified dimensionless velocity, Re_D , that varies parametrically up to a nominal Reynolds number,

$$Re_{D,in} = \frac{\rho v_{in} D}{\eta}$$

The outlet was set to Outlet, Pressure, no viscous stress, $p_0 = 0$. The wall was set to no slip and the centerline to axisymmetric. The mesh for the 2-D axisymmetric solution should be as fine as needed for an accurate solution because the mapped solution will only be as good as the initial solution that it is given as a source.

For the full 3-D model, the inlet temperature is specified at 105°C. The outlet of the flow channel is given a convective flux condition. The velocity field is specified as it relates to the extruded coupled variables as previously described. The chip is given a volumetric

generation 1.78e9 W/m³. The heat generation was determined as the peak load from previous tests [3]. All other boundaries were left as thermally insulated. These boundary conditions represent a conservative evaluation of thermal performance.

In reality, some heat will be removed from these surfaces by natural convection, impinging flow, and conduction to a manifold connection. If the predicted temperatures in the thermally insulated case meet the design specifications, then the actual performance will be better (i.e., lower chip and fluid temperatures).

3. Results and Discussion

For a nominal Re^* , where $Re^* = Re_D/Re_{D,in}$, the maximum interface and fluid temperatures are shown in Figure 3. The interface temperature is the maximum temperature between the chip and the bonded copper, at point A shown in Figure 2. The maximum fluid temperature is found on the channel wall closest to the heat source, at point B in Figure 2.

For the three ceramics with higher thermal conductivity in Figure 3, the interface temperature is below the operation limit of 150°C, but the fluid temperature is above the boiling point of the working fluid, which is 130°C. The lower thermal conductivity ceramic drastically exceeds both design limits. Boiling is not desirable because it could eventually lead to localized hot spots that could damage the electronic component. Furthermore, the model was not created to simulate the phase transition at boiling, thus the temperature results are not accurate when the boiling point is exceeded. An increase in the Reynolds number will eventually bring the fluid temperature below the boiling point where modeling results can be more reliable. This effect is shown in Figure 4 for ceramic 4. Note that Re^* has to be greater than 2.6 to produce fluid temperatures within the operational limits.

In Figure 4, the pressure drop is also shown to be small for the given geometry. The pressure drop results coincide well with analytical predictions based on correlations by Shah [4] for pressure drop in laminar entry regions and correlations by Phillips [5] in the turbulent region.

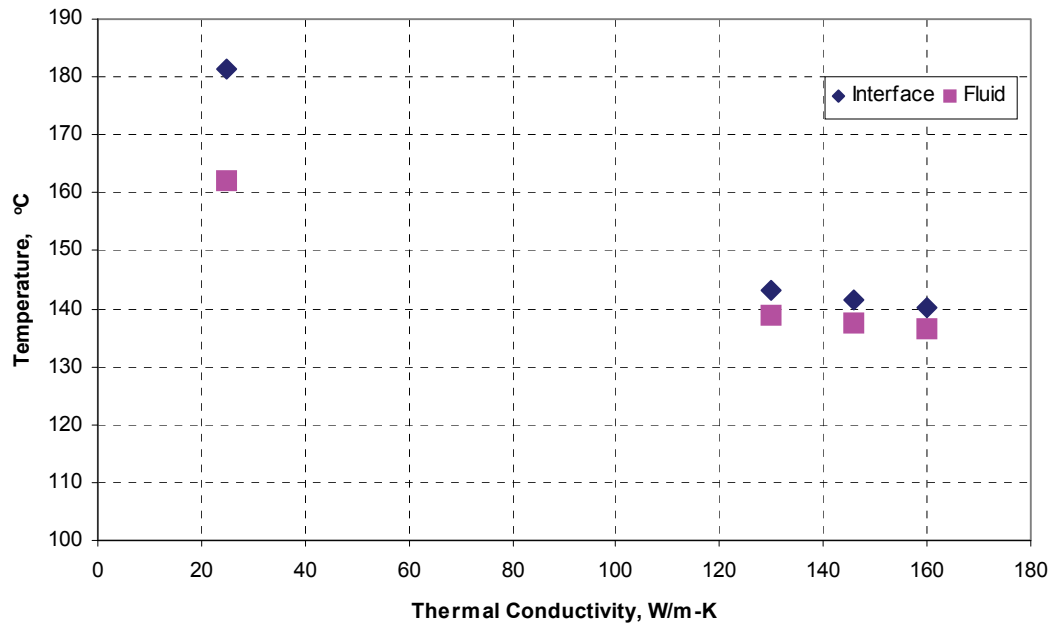


Figure 3. Maximum Interface and Fluid Temperatures of the Four Ceramic Materials in Table 1 at $Re^* = 1$

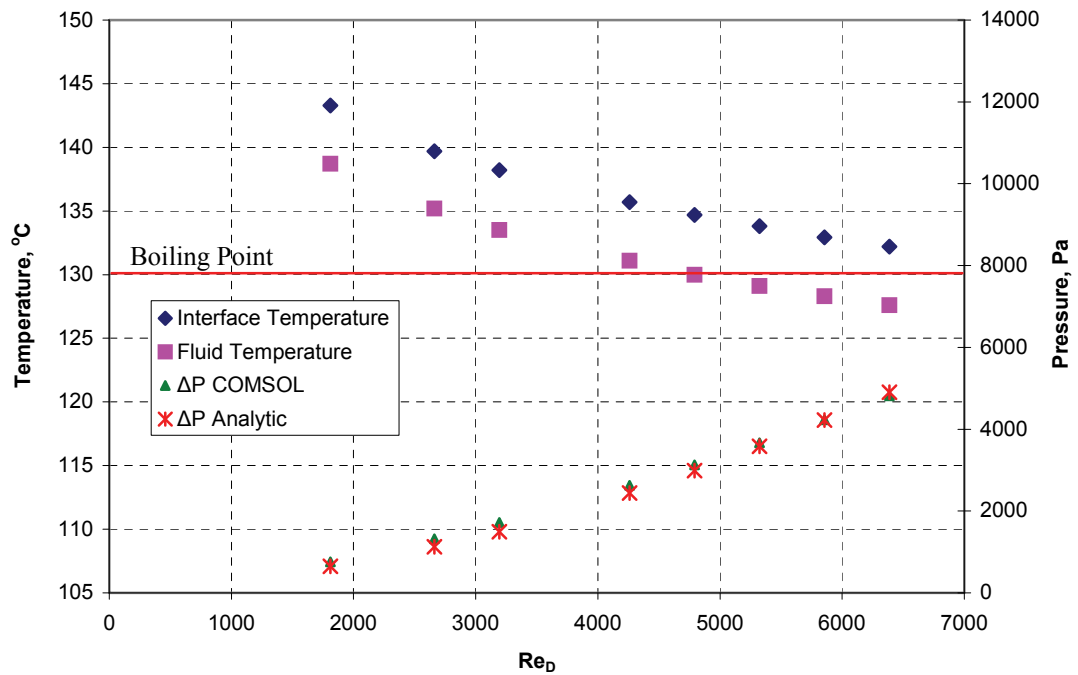


Figure 4. Variations of Maximum Interface and Fluid Temperatures and the Pressure for Ceramic 4

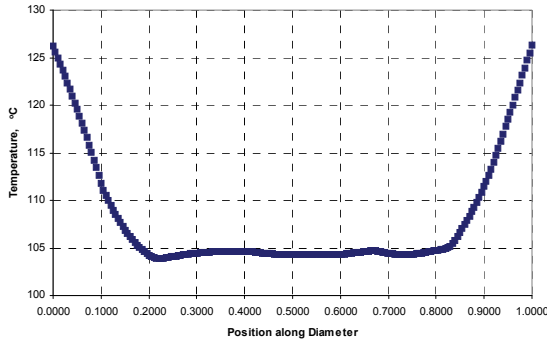


Figure 5. Fluid Temperature Along Diameter at Outlet of Hottest Channel in Ceramic 4

Though the pressure drop is small, the pressure balance between each channel is important. The large number of holes require that the pressure drop be the same for all channels to ensure even flow distribution. Uneven heating or poor manifold design would complicate the implementation of this design and could result in device damage or failure. More modeling is required to test the robustness of the design against non-uniform flow distribution at the inlet.

To move toward simpler designs with fewer flow channels, the thermal conductivity of the working fluid must be enhanced. Examining the temperature distribution along the diameter of the channel outlet shows that the thermal boundary layer thickness is small. The temperature distribution in Figure 5 shows that 41.3% of the fluid volume remains at the inlet temperature. Ideally less fluid would remain at the inlet temperature to better utilize the heat capacity of the working fluid. In this case, manufacturing and cost limitations prevent interior grooves or fins from being integrated into the ceramic core. However, thermal conductivity enhancers could be implemented. Even small increases in the effective thermal conductivity of the working fluid would provide dramatic improvements because the thermal conductivity of the working fluid is two orders of magnitude less than the lowest value of thermal conductivity of the ceramic candidates.

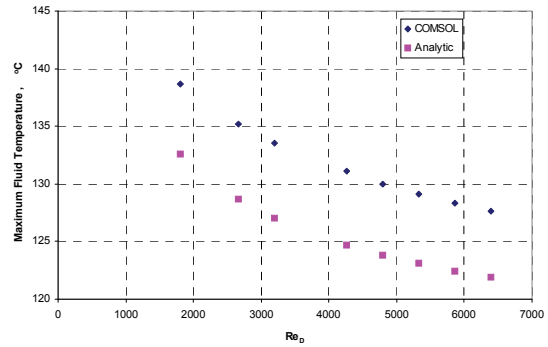


Figure 6. Comparison of Maximum Fluid Temperature between COMSOL Solutions and Analytical Solution for a Constant Flux Tube

4. Solution Accuracy

Developing three-dimensional models is a great tool to better understand the heat flow and temperature distribution in heat exchangers. However, the time investment must be warranted. Furthermore, it is unrealistic to model all appropriate design configurations. In this case, the flow channel shape is simple, and the COMSOL results can be compared to analytical solutions for channel flow with a constant heat flux wall. Correlations given by Churchill and Ozoe [6] for heat transfer in developing flows best fit the application at hand.

Figure 6 shows the predicted maximum fluid temperature versus the COMSOL solutions. The analytical solution is expected to be less than the COMSOL predictions because the analytic solution uses an uniformly distributed heat flux, where as in the real case the heat is concentrated to a smaller area and is located only on one side of the channel. The error between the two methods is less than 5%. Thus rough sizing of channel size, length, and fluid flow parameters can be done with a reasonable level of confidence prior to investing in the modeling procedure.

5. Conclusions

To provide better cooling for power electronic devices, a new direct bonded copper substrate is proposed. This substrate is actively cooled which eliminates some of the largest thermal resistances typically encountered in power electronic packaging. In order to simulate the temperature distribution in the design, COMSOL was used to develop an efficient method of solution mapping to drastically decrease the total computational time required without loss of solution accuracy.

The results showed that the chip receives adequate cooling at low flow rates, but the flow rate must be increased so that the fluid temperature does not exceed the boiling point of the working fluid. For general designs, high thermal conductivity ceramics are desirable as they provide lower interface and fluid temperatures. At large flow rates, the thermal boundary layer is thin, and a large core of cold fluid exits the fluid channel. Design improvements could be made to transfer more heat to the fluid by enhancing the thermal conductivity of the working fluid or redesigning the flow channels to provide more wetted surface area. Research is being continued to explore flow channel pressure balance, design robustness, and thermal conductivity enhancements.

6. Acknowledgements

Prepared by Oak Ridge National Laboratory, managed by UTBattelle, LLC, for the U.S. Department of Energy under contract DE-AC05-00OR22725. The submitted manuscript has been authored by a contractor of the U.S. Government under contract DE-AC05-00OR22725.

Accordingly, the U.S. Government retains a nonexclusive, royalty-free license to publish or reproduce the published form of this contribution, or allow others to do so, for U.S. Government purposes.

7. References

1. Randy H. Wiles, Andrew A. Wereszczak, Curtis W. Ayers, and Kirk T. Lowe, Direct Cooled Power Electronics Substrate, provisional patent number 61/037,129, March 17, 2008
2. 2D Axisymmetric Transient Heat Transfer, *COMSOL Multiphysics Modeling Guide Version 3.4*, 188-194. (2007)
3. S.C. Nelson and C. W. Ayers, Testing of the Semikron Validation AIPM Unit at the Oak Ridge National Laboratory, ORNL/TM-2005/44 (2005)
4. Frank M. White, *Viscous Fluid Flow*, 289. McGraw Hill, Boston (2006)
5. Frank P. Incropera, *Liquid Cooling of Electronic Devices by Single-Phase Convection*, 79. John Wiley & Sons, Inc, New York (1999)
6. Adrian Bejan, *Convection Heat Transfer*, 3rd edition, 136. John Wiley & Sons, New Jersey (2004)