Advanced Power Electronics—Thermal Management

Desikan Bharathan, Keith Gawlik, Dr. Bill Kramer
National Renewable Energy Laboratory
1617 Cole Boulevard
Golden, Colorado 80401
Voice: (303)-384-7418; Fax (303)-384-7495; E-mail: Desikan_Bharathan@nrel.gov

DOE Program Manager: Susan Rogers
Voice: (202) 586-8997; Fax (202) 586-4403; E-mail: Susan.Rogers@EE.DOE.GOV

NREL Task Leader & Technical Advisor: Dr. Terry J. Hendricks
Voice: (303) 275-4419; Fax (303) 275-4415; E-mail: terry_hendricks@nrel.gov

---

Objectives

Develop advanced thermal management methods and systems that will allow next-generation power electronics to operate at high heat fluxes and high temperatures in a compact (low volume), lightweight power electronics package.

Approach

- Analyze the cooling and thermal control technology currently used in state-of-the-art insulated gate bipolar transistors (IGBTs) for high power applications, such as in automotive traction drives.
- Explore variety of improved options for cooling within given manufacturing constraints.
- Provide recommendations to manufacturers to accommodate improved flow and thermal behavior of cooling systems.
- Test and validate improvements when prototypes are available.

Accomplishments

- A state-of-the-art inverter liquid cooled heat exchanger was redesigned using a combination of computational fluid dynamics (CFD) and thermal modeling. Analysis predicts that the new geometry will reduce the temperature of the cold plate at the junction by 8°C. This could allow the inlet coolant temperature to be increased from 70°C to 78°C. The analysis also indicates that the new design maintains a high coolant velocity throughout the flow field thus making the plate less prone to fouling.
- An air-cooled heat sink proposed by researchers at the University of Arkansas CARAT Program was modeled and analyzed. The power assembly uses 12 sets of MOSFET chips that generate about 65 W of heat distributed highly non-uniformly. The heat sink design was analyzed with and without aluminum extruded fins for heat rejection to air. With the selection of a particular commercially available extrusion, the heat transfer modeling and analysis predicts that the chip temperatures can be maintained at less than a high limit of 125°C. The research is described in the University of Arkansas's Annual Report.

Future Direction

- Develop an advanced inverter heat exchanger using advanced cooling concepts (e.g., advanced heat pipes) and advanced FLUENT thermal/fluid dynamic models. Validate thermal/fluid models using test data.
- Perform analysis on various spray and jet configurations, flow rates, and spray patterns, as well as on the use of multiple jet arrangements, and the effects of varying the distances from the jets to the target surface. Conduct research on the effect of surface preparations, such as etching, dimpling, and patterning the underside of the die or substrate, on spray cooling performance and increasing the thermal conductance between the semiconductor die or substrate and potential cooling fluids.
- Perform in-depth research on advanced heat exchanger concepts that may advance near term inverter designs including ability to accommodate variable heat loads, variable pressure conditions, and vibration conditions in future advanced vehicles according to industry specifications.
- Perform vehicle-level co-simulations of thermal management technologies using a combination of ADVISOR and component-level simulations.
- Working closely with industry partners, NREL will look for opportunities to identify techniques for economical packaging of advanced, next-generation power electronics, and ways to mitigate failure modes or create graceful degradations upon failure.

Introduction

One critical barrier to developing next-generation power electronics is cooling at high heat fluxes (~100 Watts/cm^2) at high temperatures (>105°C) in compact (low volume), lightweight power electronics packages. High temperatures at the electronic chips not only decrease their life time by accelerating failure mechanisms in materials, but reduce the overall reliability of the assembly by accelerating failure mechanisms in connections and interfaces. To overcome the barriers and challenges in next-generation power electronics cooling, it will be necessary to use advanced heat transfer and cooling techniques. The focus of this research is to improve the heat transfer capability of an existing design for an electronic assembly by changing the geometry and pin orientation of the heat exchanger. Combined computational fluid dynamics (CFD) and heat transfer simulation analysis suggests that, by using oval geometries for the pins and module mounting bosses, the inlet fluid condition to the pin fin heat exchanger can be increased from approximately 70°C to 78°C and at the same time fouling can be reduced. Efforts are underway to incorporate the suggested alterations to the design and verify the improved performance in controlled experiments.

Approach

In August 2003, NREL began evaluating a heat exchanger design that was used in conjunction with a state-of-the-art inverter that is currently being developed for DOE and the automobile industry. Analysis of the preliminary design using FLUENT, a CFD software package, showed substantial flow mal-distribution of the coolant in the entrance region, coolant flow bypass along the walls of the plate and flow detachments from the fins that limited the heat transfer, and many regions of wakes and flow re-circulation that could potentially cause fouling.

A set of proposed geometries were analyzed that could overcome the stated problems of the original design. The inverter manufacturer evaluated the suggested changes as to their practicality and manufacturability. A new, enhanced-performance inverter heat exchanger was developed using a combination of CFD

and thermal modeling. A prototype heat exchanger based on the model is currently being built. The prototype will be tested and the test data will be used to validate the model's temperature predictions.

Figure 1 shows a plan view of the cooling plate heat exchanger geometry near the coolant entrance region. The flow channel is nominally 42 mm wide, 9.6 mm deep and approximately 300 mm in length. IGBTs are mounted on the backside of the cooling plate. A thermal flux of approximately 40 W/cm^2 is expected at the worst operating conditions. The heat exchanger is made of cast aluminum. The coolant enters the flow channel via a 9.5 mm diameter inlet circular tube. The heat exchanger is made up of pin fins projecting into the flow channel to dissipate and transfer the heat to the coolant. Each pin is nominally 2 mm in diameter and 8.6 mm in length. Pins are oriented in a staggered manner with 8 and 7 pins in parallel rows. The rows are nominally 5 mm apart.

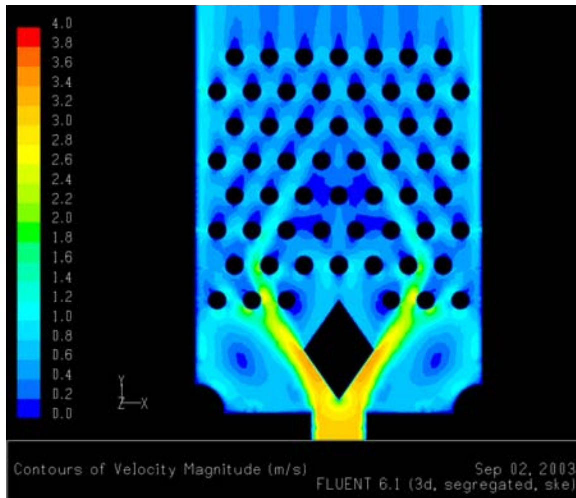

Figure 1. Flow field and velocity distribution at the entrance region of the existing design.

To mount the electronic components on the cooling plate a set of four mounting holes are drilled and tapped from the backside of the plate. The diamond boss seen in Figure 1 at the entry represents the location of one such hole. The locations of these mounting holes are fixed due to assembly design constraints.

Figure 1 shows the velocity distribution in the current design. A nominal design coolant flow rate of 10 l/min is used to generate the resulting flow field. CFD analysis shows how the incoming flow impinges on the diamond boss and causes the flow to split into two major streams. These narrow streams then impinge directly on the pin fins in the first few rows. Because of the high impingement velocity at the diamond boss, the flow attaches to the sidewalls of the boss and a substantial portion of the flow is diverted around the interior pin fins. The CFD analysis predicts two large re-circulation regions at the entry and a large wake downstream of the boss. Consequently, the analysis shows that a major portion of the coolant bypasses most of the interior fins and flows along the two straight unobstructed channels along the sidewalls, thereby decreasing the heat exchanger effectiveness and increasing its thermal resistance in the inverter cooling design. This inevitably causes IGBT temperatures to increase.

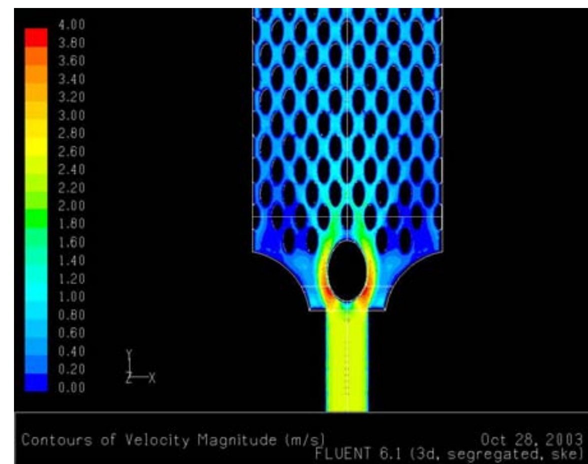

Figure 2. Flow field and velocity distribution in the revised design with elliptical boss at entry and elliptical fins

NREL analyzed a variety of potential geometries for the flow channel and illustrated what can be improved from the current design. After

consulting with the manufacturers, the geometry shown in Figure 2 was agreed upon as a compromise between manufacturability and potential performance.

Figure 2 illustrates that the flow around the boss now remains attached and smoothly distributes to the fins downstream. The reduction in the flow channel width at entry allows the flow to diffuse out more gradually. The presence of partial fins along the sidewalls as projections into the channel prevents the flow from bypassing the fins. A more uniform distribution of the flow velocity is apparent in this figure. Closer examination of the flow field reveals that the regions of flow separation behind the mounting bosses and fins are reduced substantially from the original design. The flow maintains a high velocity adjacent to the sidewalls of the pin. Both these features will keep the flow channel swept of potential debris and minimize the tendency for fouling.

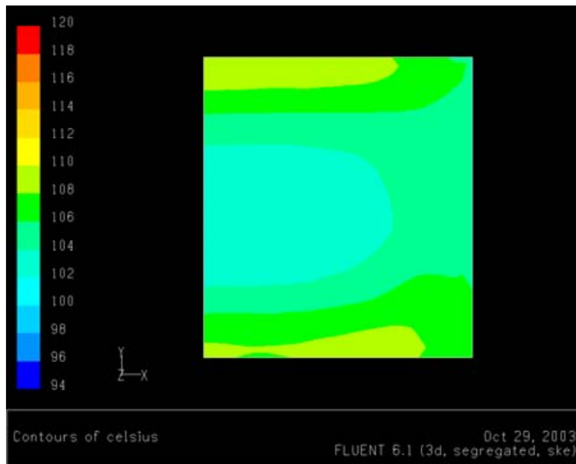

Figure 3. Plate temeprature distribution directly above a set of 4 rows of pins in the baseline design.

Figures 3 and 4 illustrate the improvements to heat transfer that resulted from the original design compared to the new design. For generating the heat transfer results, we modeled a set of four rows of fins. The geometry allowed us to treat the centerline as a symmetry boundary condition. The flow was distributed evenly across the entire channel and impinged the first row of pins straight on. A heat flux of 40 W/cm^2 was imposed at the backside of the cooling plate. Nominal coolant flow rate of 10 l/min was used for the simulation.

Figure 3 shows the temperature distribution at the heat source plate. In this picture only half of the channel width is modeled. The centerline for the cooling plate is on the left hand side of the picture. The figure shows that the plate reached a minimum temperature near the middle of the plate. This is the area where the four rows are pins are directly underneath. The model predicts that the surface temperature near the pins will be approximately 102°C and shows that the pins underneath this area are effective in tranferring the heat efficiently to help lower the plate temperature in this region.

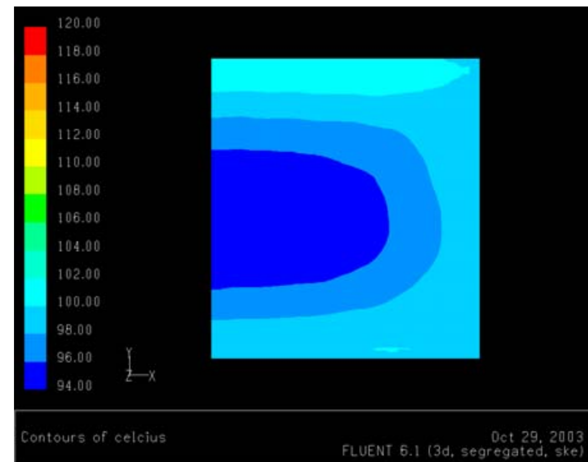

Figure 4. Plate temperaure distribution directly above a set of 4 rows of pins as configured in the proposed new design.

Figure 4 shows a similar temperature variation at the source plate for the new design, but with lower temperatures. The temperature scale is the same as in Figure 3. The temperature contours show

features similar to the ones shown in Figure 3. However, with the new design a substantial reduction in the source plate temperature can be readily observed. The model predicts that the surface temperature near the pins will be reduced to approximately 94°C.

Conclusions

A combination of CFD and thermal modeling techniques were used to develop a new, enhanced-performance heat exchanger design by changing the geometry and pin orientation. Computer analysis predicts that the new heat exchanger design will reduce the temperature of the cold plate at the junction by 8°C. This reduction could allow the inlet coolant temperature to be increased from 70°C to 78°C. Key aspects of the new design are:

- Elimination of the flow bypass that occurred along the side walls of the plate;
- Improved flow distribution in the entrance region (by redesigning the boss which is used to secure the semiconductor assembly to the cooling plate);
- Improved flow distribution around each of the pins to maintain a reasonable high velocity adjacent to the heat-transferring surfaces;
- Improved general flow distribution in the plate that eliminates major wake areas and stagnation regions. The redesigned plate is thus substantially less prone to fouling, because steady sweeping of the surface by the fluid occurs in this geometry.
- Improved heat exchanger thermal effectiveness (i.e., conductance) that results 8°C cooler source (i.e. cold) plate temperatures.

Our CFD efforts can be extended to evaluate the entire package temepratute distribution by incorporating the solid model details and conducting a conjugate heat transfer analyses in the future. CFD and thermal analyses such as these, as well as the lessons learned in this work, will be used in future work to help minimize inverter package volume and mass for automotive power electronics applications.