

2. Thermal Control for Power Electronics and Motors

2.1 Thermal Control for Inverters and Motors

Principal Investigator: John Hsu, Curtis Ayers, Chester Coomer
Oak Ridge National Laboratory
National Transportation Research Center
2360 Cherahala Boulevard
Knoxville, TN 37932
Voice: 865-946-1325; Fax: 865-946-1210; E-mail hsujs@ornl.gov

DOE Technology Development Manager: Susan A. Rogers
Voice: 202-586-8997; Fax: 202-586-1600; E-mail: Susan.Rogers@ee.doe.gov

ORNL Program Manager: Mitch Olszewski
Voice: 865-946-1350; Fax: 865-946-1262; E-mail: olszewskim@ornl.gov

Objectives

The objective of the Thermal Control for Inverters and Motors project is to reduce the size of the inverter by 50% in FY 2006 (based on the current size of the SemiKron 400-A, 160-kW inverter) and then reduce it to one-third of the original size in FY 2007 without negatively impacting the cost, life expectancy, and performance of the entire system.

Motor cooling is also addressed in this project. A cooler motor is particularly needed for plug-in hybrid vehicles because the demand on the motor is higher for providing sufficient torque and power without constantly relying on the help of the engine.

Approach

Much effort has been exerted within the electronics research community to improve semi-conductor die cooling, but little or no attention has been given to improving cooling for dc-link bus capacitors. Oak Ridge National Laboratory (ORNL) has recognized the importance of cooling all the components in the traction drive system, especially the dc-link capacitor. In this project, ORNL uses the passenger air conditioning (AC) coolant to aid in the cooling of the power electronics circuitry. Tapping into the AC system can enable cooling of the insulated gate bipolar transistors (IGBTs), diodes, dc-link capacitor, digital signal processors (DSPs), gate drives, and control power supply without relying on the AC compressor. Note that not all the components need to be submerged in the liquid refrigerant, because the refrigerant vapor and the hermetic housing of the refrigerant provide separate cooling environments for the different components of the inverter/motor drive system.

As part of this project, an oil-spray cooling method was investigated to improve motor cooling.

Approach in FY 2006

- Design and test motor heat exchanger with oil spray
- Design and fabricate hermetic container
- Fabricate film capacitor with tapered metal distribution and bare metal end connections
- Mount silicon dies in prototype cascade inverter
- Assemble and test inverter
- Perform initial integration of floating loop, inverter, and motor

Major Accomplishments

1. An oil spray cooling system for the drive motor was designed, fabricated, and tested. The cooling was significantly improved compared with oil-sling cooling systems.
2. ORNL designed, built, and tested a half-size, full-power inverter. The dc test reached 600-A capability (200 A per leg), and the ac switching test achieved 250 A_{RMS} for 3-phase output.

Technical Discussion for Motor Cooling

Motor windings will heat up in operation because of the finite resistance of the wire and the amperage load. Two primary motor cooling strategies have been used in the Power Electronics and Electric Machinery Research Center laboratory. The first, the “slinger” method, is patterned after the mechanism used in the Toyota Prius electric traction motor. The second method, the “pumped-spray” method, uses an external pump to spray cooled lubricant directly on the electric windings of the motor.

The “slinger” mechanism places a disk with a few slots cut radially into the outer edge adjacent to and rotating with the motor rotor. A portion of the disk periphery is immersed into cooling lubricant in the motor lower internals (the sump) so that during motor/disk rotation, the lubricant is picked up in the slots and hurled by centrifugal force onto the other motor internals, hence the name “slinger.” This slinger arrangement has the advantage of simplicity and low cost. An obvious disadvantage is that the lubricant flow is not spread uniformly on the motor windings around the periphery of the motor internals. (However, this simple, inexpensive method may be better in some sense under low loads or short transients than a more complicated method.) A test apparatus was assembled to mirror as closely as possible the configuration of the Prius traction motor, and then operated at load to evaluate the performance of the simple slinger cooling method. The tests were conducted in 2004.

An external pump was used to spray cooling lubricant directly onto the motor windings to evaluate this more direct spray cooling method. This pumped-spray cooling method uses a dedicated external pump that takes coolant from the motor sump, passes it through a heat exchanger, and sprays the coolant through 18 nozzles distributed uniformly over the top half of the motor casing. The sprayed coolant impinges directly onto the motor stator windings, where the coolant drips off the windings back into the sump. To compare the results of the slinger tests with the pumped spray tests, the results were made as comparable as possible. They are shown in Table 1. The laboratory ambient temperature was approximately 20°C for both test series.

Table 1. Comparison of cooling methods

	Pumped spray	Slinger
Total electrical power (W)	2270	2070
Maximum winding temperature (°C)	59.2	142.0
Lubricant sump temperature (°C)	31.9	65.0
Pumping power (W)	104.0	96.0

The pumping power for the pumped-spray method was measured by the current and voltage draw of the dc motor that powered the rotary gear pump. The pumping power for the slinger method was measured by the load on the motor that powered the rotating slotted disk. Thus even though the pumped-spray method required an independent pump, a power increase of only approximately 8% was required for the increased cooling. The viscous drag and momentum transfer experienced by the rotating disk slinger method is not as effective in pumping cooling fluid as is the rotary gear pump.

Based on an inspection of Table 1, the pumped-spray method clearly reduced the maximum winding temperature by a considerable amount compared with the slinger, while removing approximately 10% more heat. This significant improvement in cooling is no doubt a result of the higher coolant flow rate and more uniform coolant distribution of the pumped-spray method. Although this method is more involved

than the simple slinger method, the results show a distinctly advantageous lower motor winding temperature.

Details of the Pumped-Spray Cooling Method

- It uses hydraulic fluid as the coolant, sprayed directly on windings and adjacent internal structure.
- The cooling subsystem uses commercially available components.
- A stainless steel brazed plate heat exchanger is used.
- Transmission oil is used.
- A gear pump is used.
- Screen filters are used.
- Multiple spray nozzles are aimed at hot spots

The motor cooling subsystem uses spray nozzles, as shown in Figure 1. The electric motor cooling test loop was designed, assembled, debugged, and tested on August 28, 2006. The test loop circulates the coolant through a heat exchanger, pumps the coolant, and sprays the cooled fluid on the motor windings. The cooling fluid is a synthetic hydraulic fluid, manufactured by Amsoil and designated as ATH. There were 12 spray nozzles on each of the two sides of the ORNL 6000-rpm BFE test motor; 3 were horizontally aimed at the excitation coil, and the remaining 9 were radially aimed at the stator windings in the upper half of the motor casing.



Figure 1. Photo of inside of motor housing showing spray nozzle locations.

For the electrical power levels tested, the motor stator windings were cooled well below the temperature limit of $\sim 180^{\circ}\text{C}$. Test results are presented in Table 2, which reports the measured pressures; Table 3, which reports the measured phase currents, voltages, and total power; and Table 4, which reports the measured temperatures. The motor was wired in a delta configuration. There were three tests at the stated power levels. The loop pressures experienced by the cooling fluid for all power levels are given in Table 2. The pump dc motor was set to an 8-V and 13-A draw.

Table 2. Loop pressures

Place in loop	Gauge pressure (psi)
Pump discharge	26.0
HX inlet	26.0
HX outlet	19.5
Motor inlet	19.5

Table 3. Motor winding electrical parameters

Maximum line current (A)	Leg A-B		Leg B-C		Leg A-C		Total power (W)
	Current (A)	Potential (V)	Current (A)	Potential (V)	Current (A)	Potential (V)	
109.0	48.9	16.3	59.5	16.0	57.5	16.4	1535
205.4	86.0	27.9	109.7	27.2	105.3	27.1	6970
266.0	103.9	33.6	138.0	32.3	134.2	33.7	9439

Table 4. Temperatures

Thermocouple location	Test at 1448 W	Test at 5860 W	Test at 7491 W
1. Oil sump	26.0°C	32.8°C	42.0°C
2. Motor front, 1:00 outside stator	42.3°C	91.7°C	133°C
3. Motor back, 10:00 outside stator	34.5°C	60.0°C	81.2°C
4. Motor back, 12:00 inside stator	43.0°C	81.6°C	133°C
5. Motor front, 9:00 inside stator	34.2°C	77.5°C	79.0°C

The tests were conducted by first electrically powering the windings to yield the approximate maximum line current shown in Table 3 with the hydraulic pump off. The maximum line current is seen in the third line for the delta leg configuration given in Table 3. After less than a minute, the winding temperatures started to increase noticeably, and then the hydraulic pump was energized. This heatup procedure was used to decrease the time needed for a steady state to occur. The first trial procedure had the oil flowing before the ac current was initiated, and the elapsed time to a steady-state was much longer than desired. The oil was allowed to flow until no perceived change in the winding temperatures was noticed, approximately 10–15 minutes. The ac motor current was then shut off, and the temperatures decreased quickly. The dc pump motor was shut off after a few more minutes.

Although the power supply had the capability of providing higher power, further testing at higher power was not completed because the winding fibrous insulation external to the motor case started to smoke at currents higher than the maximum reported in Table 3. This smoke was probably from residual cooling fluid on the fibrous insulation. The spray cooling system was designed to cool only the inside of the motor and not the outside. Further tests could be conducted at higher powers if cooling external to the case were provided, such as simply using a fan to blow laboratory ambient air over the external motor leads.

As seen from the data in Table 3, the delta configuration was not perfectly balanced. The total power was calculated by squaring each leg current, multiplying by the leg resistance, and then adding the three leg powers. The measured resistance for Leg A–B is 0.162 ohm, for Leg B–C is 0.154 ohm, and for Leg A–C is 0.156 ohm, all measured at the laboratory ambient temperature of approximately 19°C. The resistance values were calculated based on the average temperature of the windings for the three maximum line-current cases, respectively. Table 1 shows that the coolant pumping power was 104 W for 2270 W total system power. It is important to note that for all total powers shown in Table 3, the coolant pumping power remained at 104 W.

The heat generation rate in the windings should be uniform at any position. Referring to Table 4 data, note that the maximum stator temperature occurred at the topmost positions for all tests. Also note that the winding temperature decreased as the thermocouple position became more horizontal. The spray nozzles, which had the same orifice diameter, were evenly spaced over the top half of the motor and were fed coolant from a common supply tube. This would imply that the coolant flow experienced by the windings increased with an increasingly more horizontal position. In addition, since the lower part of the stator is immersed in the oil sump, thermal conduction through the iron core and copper in the stator transports heat from the upper part of the stator to the oil sump. To lower this maximum winding temperature, it would be advantageous to bias the flow more toward the top of the motor by increasing the orifice size at

the topmost positions, decreasing the orifice size at the more horizontal positions, or perhaps eliminating the most horizontal orifice. This topmost orifice size bias might increase the other temperatures, but the objective of the spray cooling arrangement should be to decrease the maximum temperature no matter where it occurs.

Of particular note is that the temperature rise across the water side of the heat exchanger was on the order of 10°C above laboratory ambient for all tests, and the water pump head pressure was 26 psi for all tests. This low temperature rise might indicate that the heat load on the heat exchanger was low; it might also indicate that there was a fair amount of heat loss to ambient. The heat exchanger was noticeably warm to the touch but not hot, and there may have been a significant heat loss from the bare metal of the heat exchanger to ambient. The heat exchanger was oriented vertically, which might increase any natural convection to the lab ambient.

The ORNL tests described show that the hydraulic fluid motor cooling loop will keep the stator winding temperatures well below the temperature of concern for the power levels listed for these tests, even with the coolant pumping power maintained at a low level. The cost for the ORNL approach will be slightly higher because of the additional pump. However, an effort is ongoing to attempt to modify the exchanger and to incorporate the pump with the shaft to reduce costs.

Technical Discussion for Inverter Size Reduction

The goal for FY 2006 was to reduce the size of the inverter by 50%, using the Semikron inverter (Model 4001GD06) as a baseline (Figure 2).

Data published by Semikron rate this inverter at 160 kW using 25°C coolant, and 82 kW with 70°C coolant, with the *specific power* at 9.9 kW/kg and the *volumetric power density* at 9.5 kW/liter. The derated current for the 70°C condition is about 250 A_{RMS}. For purposes of comparison with the ORNL inverter that is based on a 50–60°C maximum coolant temperature (bulk liquid temp in a 2-phase system), the Semikron rating at 70°C will be used to provide a more even rating comparison. The ORNL inverter is rated at approximately 65 kW (based on dc test results and cooling capabilities), with 19 kW/kg and 21 kW/L power capabilities. Key specifications are compared in a tabular form in Table 5.

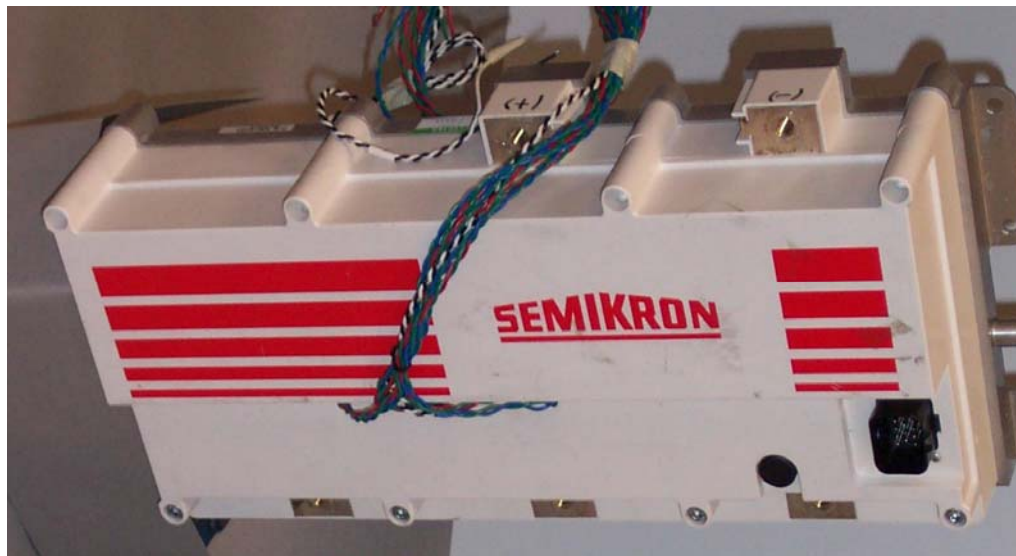


Figure 2. SEMIKRON Model 4001GD06 82-kW inverter.

Table 5. Comparison of key specifications

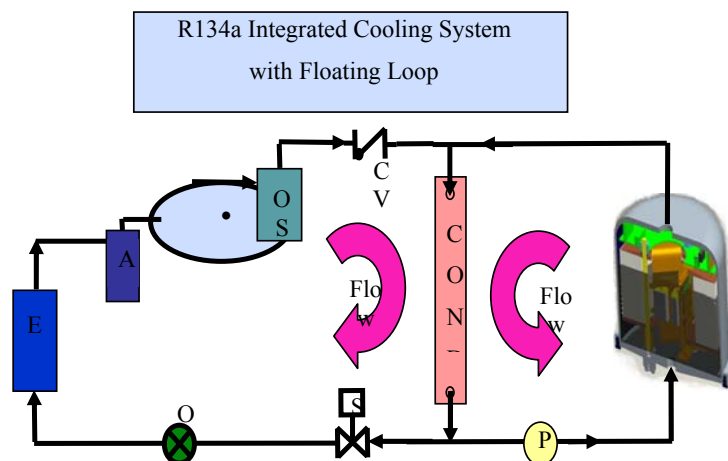
	Semikron 82 kW	ORNL 65 kW	FreedomCar targets 55 kW
Volume (l)	8.6	3.1	6.1
Full load amps	252	250	
kW/kg	9.9	19	11
kW/liter	9.5	21	9

Approach to Size Reduction

- Direct-contact 2-phase cooling technique (ORNL floating loop)
- Capacitor size reduction through shape innovation and lower-temperature environment
- Shift from conventional power electronics packaging methods

1. Direct two-phase cooling technique

The ORNL floating loop cooling system (refer to Publications section and related patents in the Patents section) is a direct contact 2-phase cooling technique that improves cooling sufficiently to partially enable a reduction in size from existing conventionally cooled inverters (Figure 3). The improved heat transfer coefficient achieved by directly submerging the power electronics devices and conductors in refrigerant (R134a in this research) allows the silicon devices to be pushed to higher current levels than conventional cooling will allow, with the same or reduced junction temperatures.

**Figure 3. ORNL floating loop schematic diagram.**

Conventional design (e.g., the Semikron design) depends on conduction cooling from one side of the silicon power chip to transfer heat through an electrically insulating layer (usually aluminum nitride) into the heat sink. The thermal resistance of this setup is high compared with direct 2-phase liquid cooling, in which the coolant itself completely surrounds the chips and the associated electrical conductors near the chips. Heat fluxes of as much as 95 W/cm^2 (using both sides of the silicon chip as heat transfer areas) have been achieved in the laboratory during testing of the ORNL FY 2006 half-size inverter. This result is based on dc load testing in which up to 167 W has been dissipated from each of 12 IGBT chips at the

peak dc loading. Similar calculations for the Semikron inverter indicate about 42 W per chip of heat dissipated, which equates to less than 50 W/cm² (using one side of the chip as conduction heat transfer area). These data indicate that about four times as much heat is being removed from each chip using the direct-cooling technique.

The current test results were limited at this stage by the loading equipment capabilities presently available at ORNL. Testing will continue for higher currents when the loading capability is increased in the near future. The gate driver circuitry, which was designed for a lower-current-capability IGBT, presently exhibits a saturation voltage protection that is too low. This situation is also limiting some of our output current tests. The protection enables around 250 A for a given leg of the 3-phase circuit, which is made up of two IGBT chips. The IGBTs themselves have been tested using a Tektronix high-power curve tracer and do not show any current saturation in this range. The saturation curve in Figure 4 shows that the pair of chips can handle a collector current of 384 A and that the V_{ce}/I_c curve is still linear at the 384-A point. This indicates that the pair of chips can exceed this value by a significant amount, allowing the chips to be pushed to a higher current than previous tests have used.

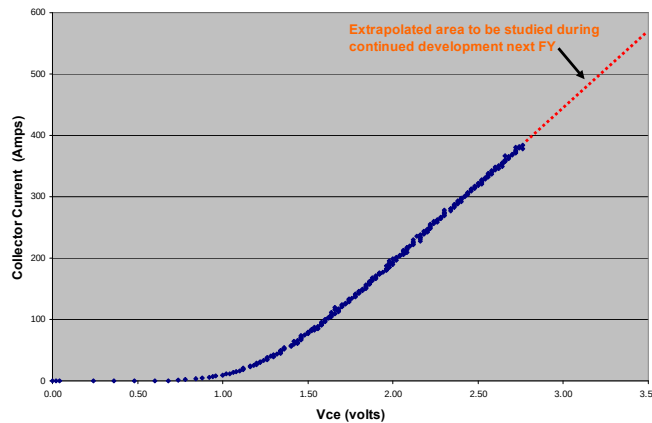


Figure 4. Curve tracer data showing Semikron current capability.

During the next fiscal year, the gate drive circuits will be fully matched to the IGBTs that are chosen for the final 1/3-size prototype. This modification will enable maximum power density to be drawn from that assembly. The loading equipment will be upgraded to handle the needed inverter current and power output to complete the full-power testing.

2. Capacitor size reduction

The dc link capacitor design that was chosen for this project has resulted in a much smaller capacitor for the ORNL cylindrical half-size inverter. The capacitors being used are polypropylene film, 900 V and 500 microfarads (μf). Polypropylene capacitor has been undergoing various types of tests while submerged in R134a over the past 2 years. The actual inverter capacitor being used has been tested in a submerged manner over the past 6 months for ripple current capability and is now being tested in the actual full-power inverter configuration in dc and ac tests. These tests prove the viability of the capacitor for this type of use and thus enable the size and volume of the required dc link capacitance to be significantly reduced. Tests to date indicate we can achieve a 50% size reduction, and ongoing ripple current tests will determine what the maximum reduction in capacitance for inverter usage can be.

Laboratory Capacitor Performance Tests

- Maximum ripple current: 300 A_{RMS}
- Resonance frequency: 8 kHz

Figure 5 shows the Semikron inverter capacitor on the left and the cylindrical inverter capacitor on the right. Not only is the configuration significantly different, but also the volume and thus the mass are very different. The Semikron capacitor depends on conduction heat transfer through the bottom of the package and acceptable ambient conditions around the top of the inverter housing. The ORNL capacitor configuration, along with the design concept, allows the capacitor to be directly surrounded by liquid coolant. This insulates it from ambient conditions to a large degree and maintains the capacitor core temperature close to bulk liquid temperature. The typical operating temperature for the capacitor in this design will be less than 60°C, as opposed to temperatures under the hood of around 90–125°C.

Capacitor comparison:

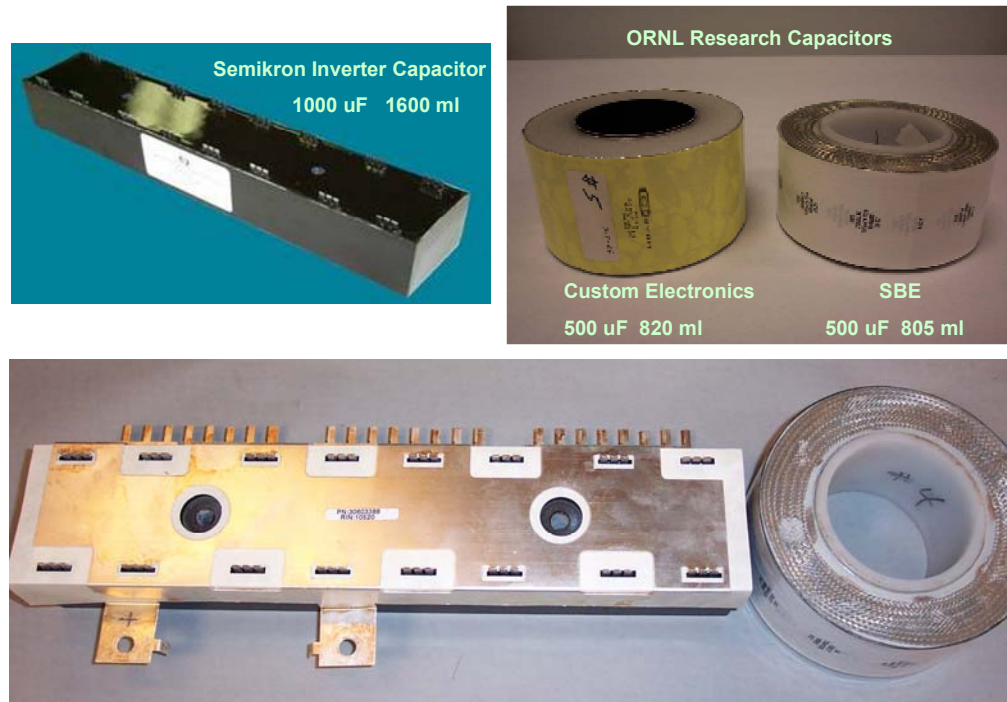


Figure 5. Comparison of Semikron 82-kW dc link capacitor and Custom Electronics and SBE cylindrical capacitors for ORNL half-size inverter. The ORNL design reduces capacitance and volume by 50%.

3. Shift from conventional electronics packaging methods

The long, rectangular block shape of the Semikron capacitor fits the conventional inverter design, which is typically planar and rectangular. Heat removal from the typical inverter is unidirectional through the bottom of the package to a cold plate cooled with water or air. The ORNL approach is to achieve the most volume-efficient shape, which is a sphere or cube. The cylindrical capacitor with a hollow core enables this type of 3-dimensional design.

The first ORNL FY 2006 prototype was developed to perform dc load testing. Figure 6 shows the core of the power section of the inverter, which is built from subsections of the Semikron DBC power electronics cards (Figure 7). The dc link capacitor was not needed for these tests, so it is not installed in this prototype.



Figure 6. Core of first FY 2006 ORNL prototype inverter.

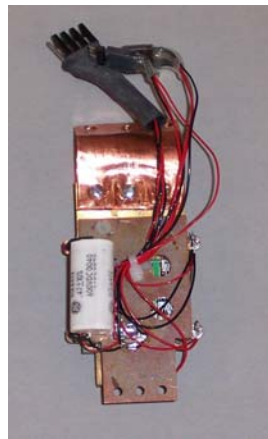


Figure 7. Power IGBT subassembly.

ORNL chose to use the Semikron DBC power board for the FY 2006 half-size inverter prototype for expediency in building and testing the whole inverter geometry to meet the FY 2006 goals. In the meantime, plans and modeling efforts are ongoing to develop the final substrate geometry that will be used to help complete the final 1/3-size inverter. The full board used by Semikron provides power to one of the three phases of the inverter. ORNL was able to cut the full board into three discrete subsections to efficiently turn one board into three boards to serve for all three phases on the new inverter. Figures 8 and 9 show the full board and the subsection used by ORNL. The subsection is the power section used in

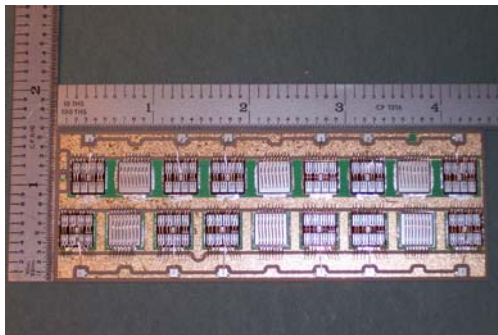


Figure 8. Semikron power IGBT board.

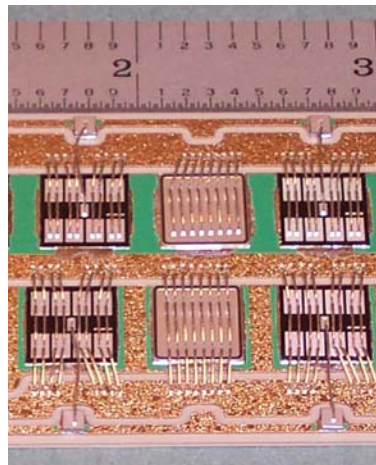


Figure 9. One of three sections of the Semikron board used to produce subassemblies for prototype.

the power subassembly shown in Figure 7. It uses 1/3 of the silicon chips required in the full Semikron board but provides essentially the same current and power performance.

The first row seen in Figure 8 makes up one of the six legs of a full inverter with an IGBT-diode-IGBT arrangement. The second row of chips makes up a second leg in one phase of the inverter—the pair can be represented by A_{upper} and A_{lower} , for example, for the A phase. The 3-dimensional shape that is the core of ORNL's design can begin to be seen in this assembly. It becomes even more evident with the final prototype half-size inverter design seen in the 3-dimensional model shown in Figure 10.

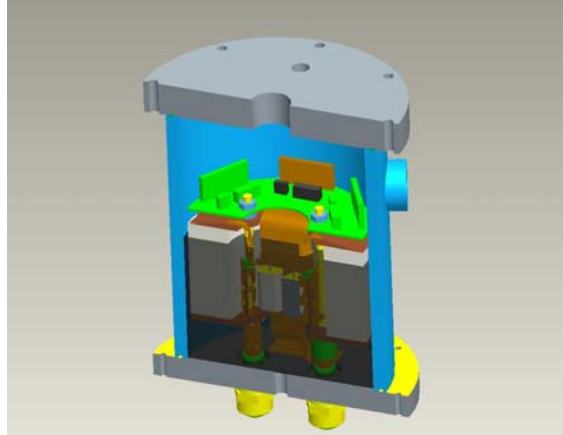


Figure 10. A three-dimensional model of the ORNL prototype showing assembly components.

In the cross-section view in Figure 10 can be seen the dc link cylindrical capacitor; the power electronics subassemblies are packaged in the hollow core of the capacitor. The dc links, disks mounted on the top and bottom ends of the capacitor, feed dc power from the external feed-throughs (seen at bottom) to the power electronics subassemblies in the center. The 3-phase output from the subassemblies exits the housing through the bottom alongside the dc input feeds. The dark shaded area shows an approximate liquid level for the coolant, and the gate driver and interface card can be seen on the top of the assembly.

Liquid feed to the system is through the bottom in the center, and vapor removal (to the condenser) is through the center of the top. This design depicts our test vessel, not a "final" design. A model of a standalone inverter might look like the unit shown in Figure 11. This model has been used to calculate the specific power and volumetric power density, and a plastic vessel/assembly has been built to verify packaging possibilities.

This model was developed with ProEngineer Wildfire software, and materials and densities of each of the parts were supplied to the software. Using this information, the software was used to provide volume and mass information for the whole model.

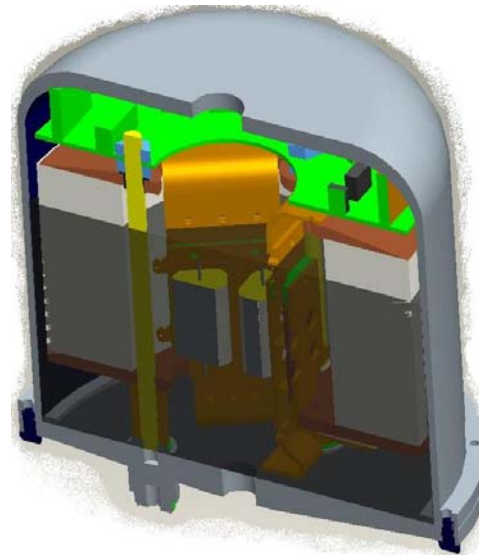


Figure 11. Model showing tighter packaging.

The FY 2006 ORNL model, which is calculated to provide approximately 65 kW of power output, has an estimated volume of 3.1 L and an estimated mass of 3.5 kg. The size and mass calculations include the DSP controller, accessory power supplies, and an interface board, estimated using the solid modeling software. For this phase of the development, these components were placed outside the pressure housing. Future developments can show the feasibility of internal or external mounting for these components. The finished inverter core is shown in Figure 12. The subassembly is complete, ready for insertion into the housing in preparation for full-power ac output. The system power is fed into a dummy R-L load on the 3-phase output to test for maximum current capability.

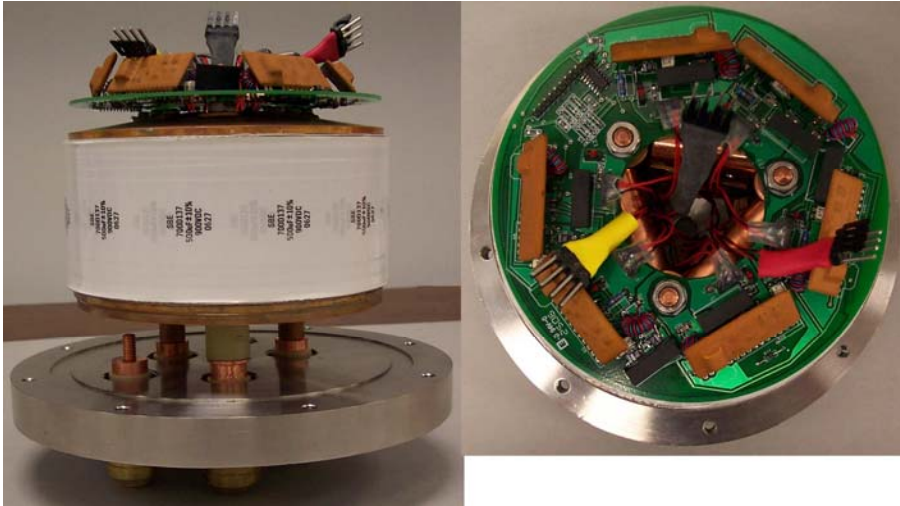


Figure 12. Side view and top view of ORNL prototype internal components.

Laboratory Testing for the ORNL Inverter Prototype

The first prototype was built early in the year to provide testing and data, using the re-sized Semikron boards. This unit primarily served as the dc loading prototype, but lessons learned in construction and mounting of the dies were used to improve the final half-size prototype design built later in the year. dc load testing was performed in the following manner. The inverter was built as a full-fledged 3-phase inverter with individual gate controls as usual. For the dc test, the plus and minus dc link were wired directly into the power supply with no load in between. The phase outputs of the inverter were left open-circuited since no ac output was being created. The gates for all the legs of the inverter were turned fully on for the duration of the test, and the chips themselves served as the load for the test in a short-circuit configuration. This configuration would cause the IGBTs to fail if the current during the test were not controlled, so the power supply was operated in current control mode. The voltage was allowed to float as the current was increased for the dc tests. This method produces heat in the IGBTs without the need to produce controlled ac output currents, and it was appropriate for the initial year's load testing. The IGBTs heated up with increasing dc current, and a temperature-to-current capability was determined. A dc test setup was developed to provide a method of measuring junction temperatures based on forward voltage drops, in order to correlate surface-mounted temperature measurement methods (such as thermistors or thermocouples). We hope to develop this method so that it can be used directly in operating inverters; but at this point, it requires a more steady-state operating condition (dc instead of ac output).

ac load testing was set up in a much more conventional manner: a full-fledged inverter with a DSP controller connected to a 3-phase R-L output load. The power was supplied to the inverter by a dc power supply at voltages up to 350 V_{dc}. The existing load equipment at ORNL at this time prevented full power

(full calculated power is 65 kW), but it reached approximately the full load current required of 250 A. The FreedomCAR inverter output capability was reached (55 kW), but higher values are expected from this inverter design that will produce the high specific power and volumetric power density that were calculated based on dc test results. Some trouble was experienced above 225 A_{RMS} , but only because of a mismatch of the gate driver and the particular IGBTs that were used for this prototype. A false saturation condition was detected, causing the gate drivers to shut down at around 250 A_{RMS} . As seen in the Semikron saturation curve (Figure 4), this value can be increased significantly (hopefully to $\sim 400 A_{RMS}$) when the gate drive problems are resolved.

The test results for this project year are listed in Table 6. Data sheets from the power analyzer taken at the various test levels are found in Appendix A. These results, along with the calculated specific power and volumetric power densities, show that the goal of a 50% reduction from the baseline for inverter size was reached.

Table 6. One-half size inverter test results

dc OUTPUT TEST (all gates turned on, current controlled):

— used to determine overall cooling capability for the IGBTs, and maximum current capability

dc link (volts)	dc current (A)	Power losses (W)
3.3	400	1430
3.6	500	2000

ac OUTPUT TEST (operated with DSP controller with R-L output load):

— these tests help determine overall operability of the inverter, and maximum power capability

dc link (v)	Modulation index	V_{ac} output	A_{RMS} output	Power out (kW)
350	0.95	255	153	55
150	0.15	25	200	4 (load reconfigured)
104	~ 0.2	45	225	8.1

Conclusions

- Tests on the oil spray cooling of the motor confirm that oil-spray cooling is a more effective cooling system than oil-sling cooling system in the same motor structure.
- Motor cooling test data are as follows:
 - Maximum power: 9.4 kW @ 266 A, 104 W pump power
 - Maximum temperature: 133°C @ 7.5 kW
- Coolant pump power can be tailored to the cooling load demand. For lower cooling demand periods, the pump speed (and thus power) can be reduced. When higher system power is needed, and thus more motor cooling, the coolant pump flow rate can be increased as needed.
- There is at least a 4.5 : 1 increase in motor heat removal with no change in the required input coolant pumping power.
- The 500- μ f dc link film capacitor functions well while submerged in refrigerant.
- The dc link capacitor tests reveal no limitations so far in handling of ripple current of up to 300 A_{rms} . These tests will continue in order to determine the maximum ripple current capability. Results of the ripple current capability tests will determine the final size for the capacitor. It appears that the capacitor can be reduced in size and volume.

- Tests on the half-size full rated inverter, including dc load and ac tests were conducted. The dc load tests confirmed that the inverter meets the full-load half-size objective. The output capability was successfully tested at up to 55 kW and is expected to be higher based on load calculations from dc test data. Loading capability limitations prevented testing at a higher power for this fiscal year.
- The size of the FY 2006 inverter is calculated at 3.1 L, including the DSP controller, accessory power supplies, interface electronics, and the inverter core itself. This is a reduction of more than 50% from the published Semikron size of 8 L.

ORNL One-half size inverter dc load testing

A _{dc} input	490
A _{RMS} output	347 (calculated)
Max power	110 kW (calculated)
Chip T	70 C
Delta T	38 C
Chip cooling	167 W/chip

ORNL One-half size inverter ac load testing

Maximum A_{RMS} output: 250
 Maximum power output: 55 kW (limited only by output loading capability)
 Inverter chip temp

- Specific power density for the FY 2006 prototype inverter = 19 kW/kg
 — Semikron baseline is 9.9 kW/kg; improvement is 92%
- Volumetric power density for the FY 2006 prototype inverter = 21 kW/liter
 — Semikron baseline is 9.5 kW/L; improvement is 121%

Future Direction

- Complete prototype design of ORNL substrate/IGBT mounting
- Redesign gate driver to match new IGBTs
- Use double-side die mounting without wire bonds
- Complete capacitor ripple current testing to determine final size/volume
- Integrate suitable components inside inverter shell
- Prototype and test new feedthrough concept for I/O connections

Publications

Curt Ayers and Kirk Lowe, "Fundamentals of a Floating Loop Concept Based on R134a Refrigerant Cooling of High Heat Flux Electronics," presented at SemiTherm22 Conference, Dallas, March 2006.

Kirk Lowe and Curt Ayers, "Operating Controls and Dynamics for Floating Refrigerant Loop for High Heat Flux Electronics," presented at SemiTherm22 Conference, Dallas, March 2006.

Patents

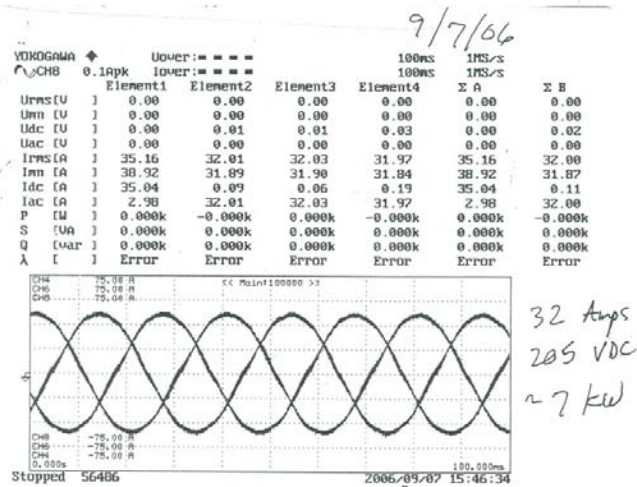
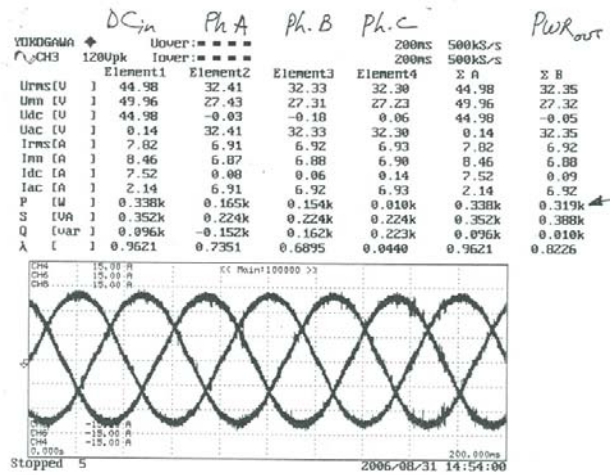
John Hsu, Curtis Ayers, Chester Coomer, and Laura Marlino, "Floating Loop System for Cooling Integrated Motors and Inverters Using Hot Liquid Refrigerant," U.S. Patent 6,993,924 B2, February 7, 2006.

John S. Hsu, Donald J. Adams, Gui-Jia Su, Laura D. Marlino, Curtis W. Ayers, and Chester Coomer, "Cascaded Die Mountings with Spring-Loaded Contact-bond Options," U.S. Patent 6,930,385, August 16, 2005.

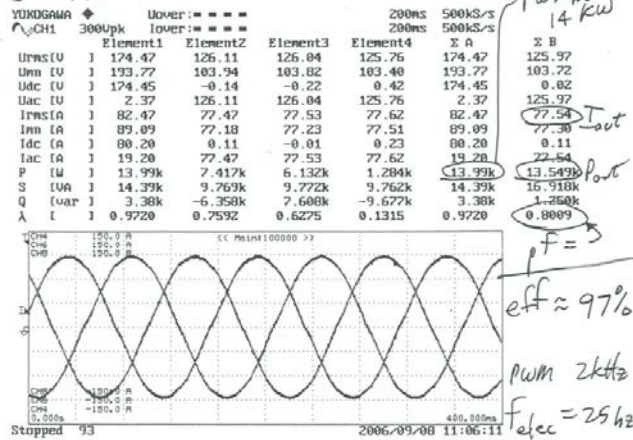
John S. Hsu, Donald J. Adams, Gui-Jia Su, Laura D. Marlino, Curtis W. Ayers, and Chester Coomer, "Total Thermal Management System for Hybrid and Full Electric Vehicles," U.S. Patent 6,772,603, August 10, 2004.

John S. Hsu and William S. Schwenterly, "Superconducting PM Undiffused Machines with Stationary Superconducting Coils," U.S. Patent No. 6,700,297, March 2, 2004.

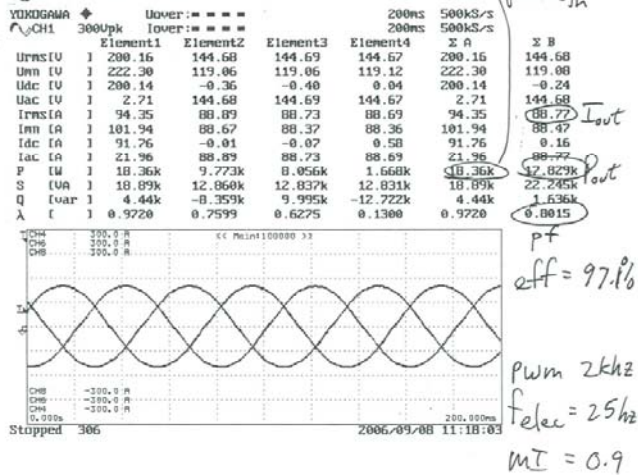
Appendix A. Power Analyzer Data Sheets from ac Testing of the ORNL Half-size Inverter



DC = 175 VDC



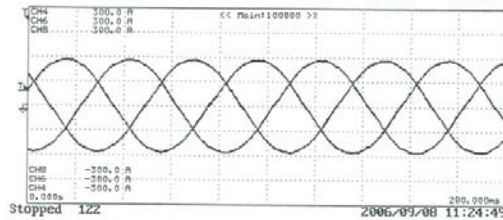
DC = 200 VDC



DC = 225 VDC

YOKOGAWA
CH1 300Upk
Lower:====
Lower:====

	Element1	Element2	Element3	Element4	Σ A	Σ B
Urms[V]	225.06	163.32	162.87	162.63	225.06	162.94
Urm[V]	249.96	134.63	133.96	133.68	249.96	134.09
Udc[V]	225.04	0.10	-0.29	0.75	225.04	0.19
Uac[V]	3.15	163.32	162.87	162.63	3.15	162.94
Irms[A]	105.86	99.96	99.58	99.00	105.86	99.78
Irm[A]	114.41	99.79	99.25	99.58	114.41	99.54
Idc[A]	103.00	0.14	-0.13	0.19	103.00	0.07
Iac[A]	24.45	99.96	99.58	99.00	24.45	99.78
P[W]	23.17k	12.419k	10.189k	2.093k	23.17k	22.587k
S[VAr]	23.83k	16.326k	16.218k	16.230k	23.83k	20.150k
Q[VAr]	5.55k	-10.598k	12.618k	-16.094k	5.55k	-2.049k
λ	0.9724	0.7607	0.6282	0.1290	0.9724	0.8028

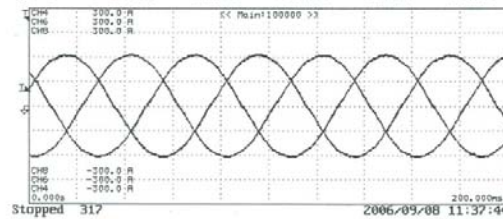


PWR in
PWR out
PWR
eff = 97.6%

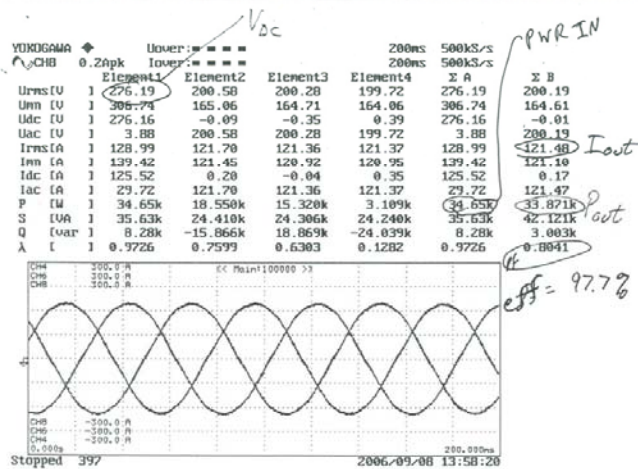
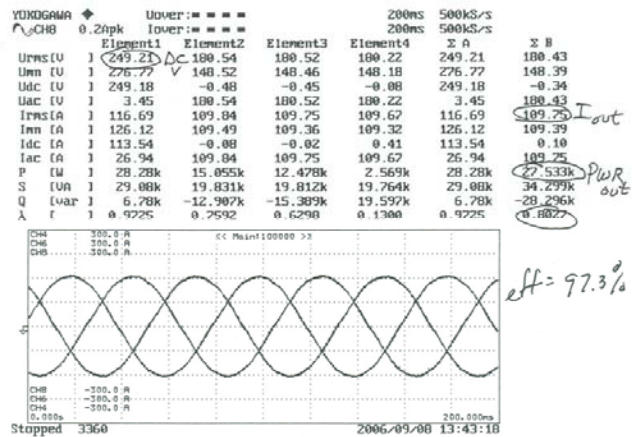
DC = 250 VDC

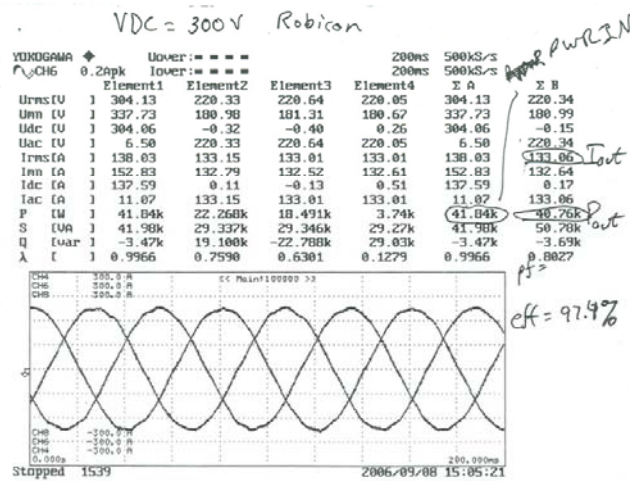
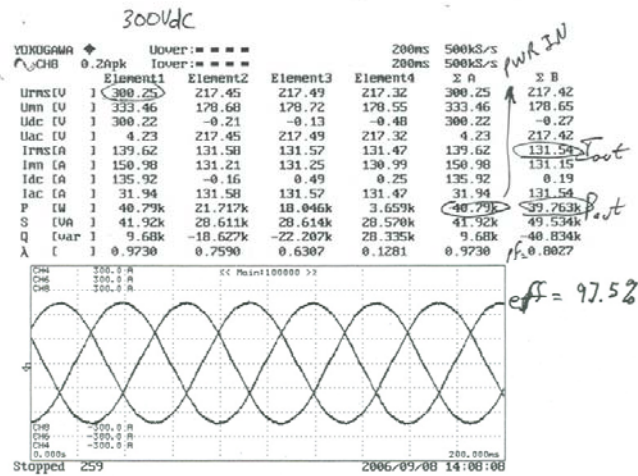
YOKOGAWA
CHB 0.44pk
Lower:====
Lower:====

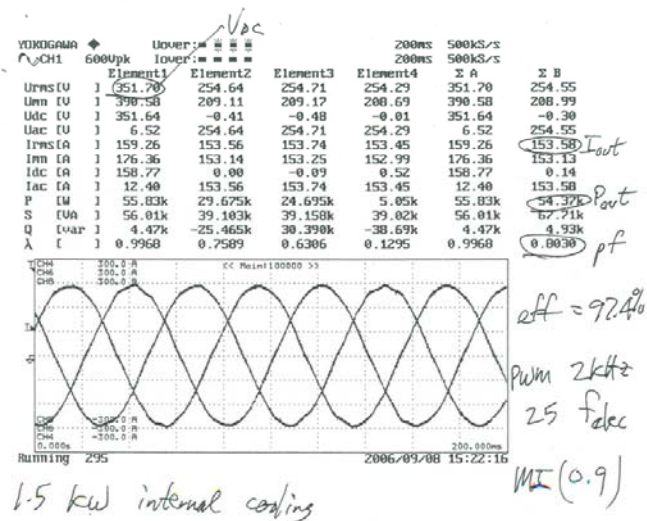
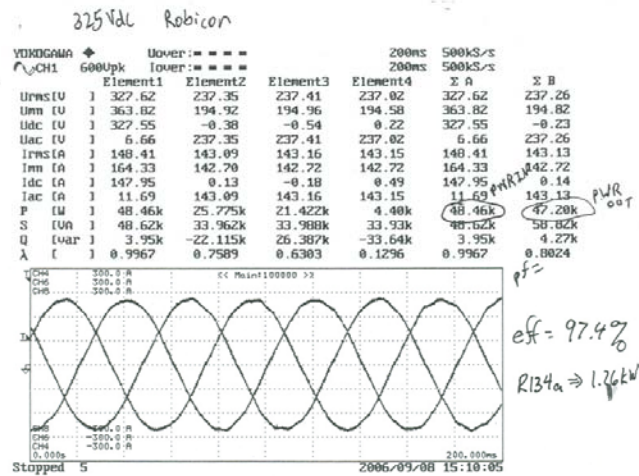
	Element1	Element2	Element3	Element4	Σ A	Σ B
Urms[V]	249.56	181.20	180.66	180.45	249.56	180.77
Urm[V]	277.16	149.19	148.50	148.21	277.16	148.64
Udc[V]	249.53	0.19	0.06	0.24	249.53	0.16
Uac[V]	3.49	181.20	180.66	180.45	3.49	180.77
Irms[A]	117.06	110.65	109.96	110.09	117.06	110.23
Irm[A]	126.51	110.37	109.48	109.68	126.51	109.64
Idc[A]	113.89	0.28	0.11	0.22	113.89	0.20
Iac[A]	27.06	110.64	109.96	110.09	27.06	110.23
P[W]	28.41k	15.255k	12.471k	2.49k	28.41k	27.73k
S[VAr]	29.21k	20.049k	19.864k	19.87k	29.21k	34.51k
Q[VAr]	6.82k	-13.009k	15.462k	-19.71k	6.82k	2.45k
λ	0.9724	0.7609	0.6278	0.1251	0.9724	0.8034

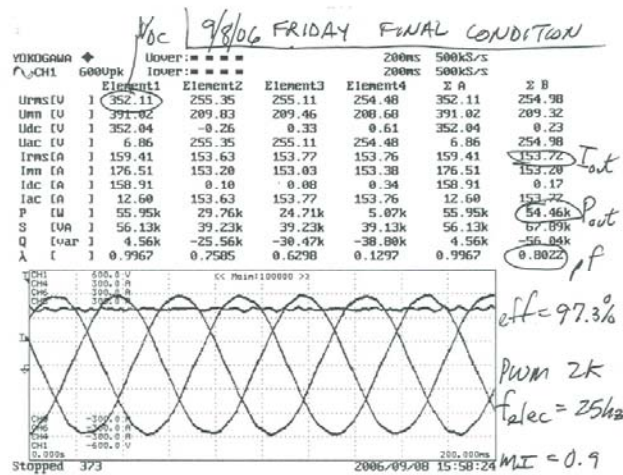


PWR in
PWR out
PWR
eff = 97.6%

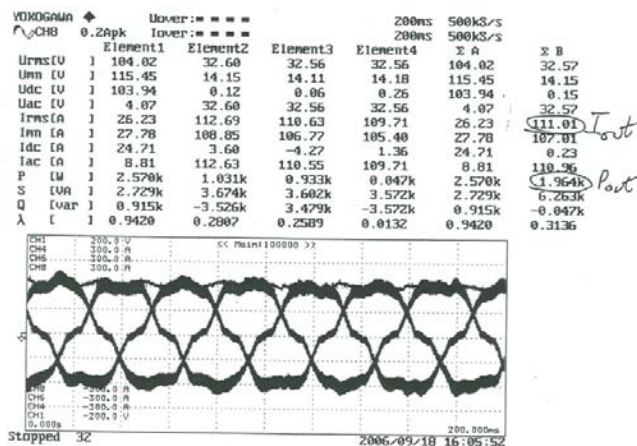
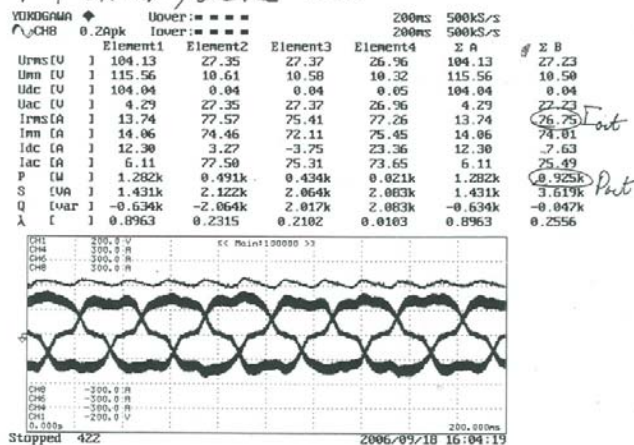


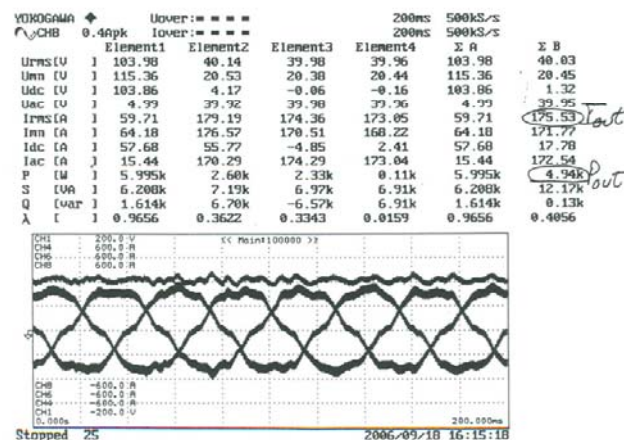
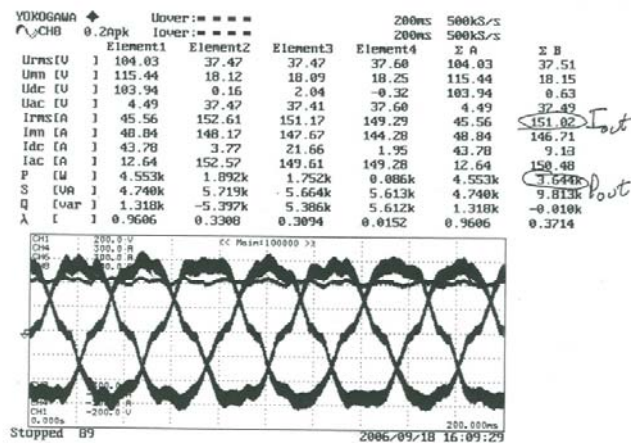


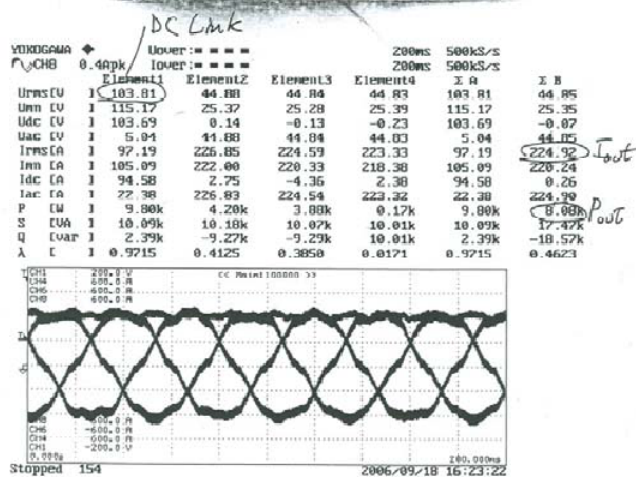
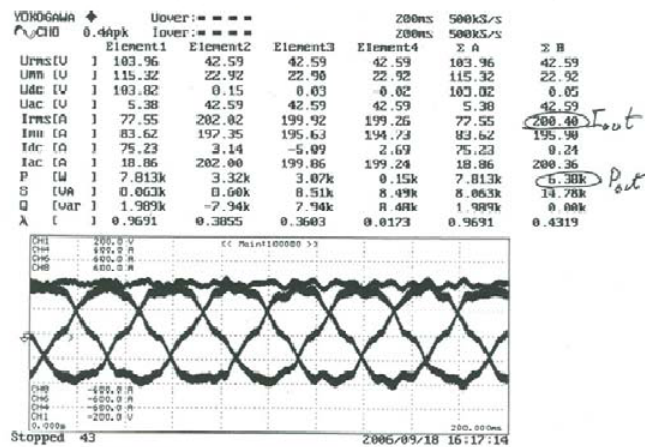




Y-Y txfmr, delta load







2.2 Identifying the Barriers and Approaches to Achieving High-Temperature Coolants

Principal Investigator: John Hsu

Oak Ridge National Laboratory

National Transportation Research Center

2360 Cherahala Boulevard

Knoxville, TN 37932

Voice: 865-946-1325; Fax: 865-946-1262; E-mail: hsujs@ornl.gov

DOE Technology Development Manager: Susan A. Rogers

Voice: 202-586-8997; Fax: 202-586-1600; E-mail: Susan.Rogers@ee.doe.gov

ORNL Program Manager: Mitch Olszewski

Voice: 865-946-1350; Fax: 865-946-1262; E-mail: olszewskim@ornl.gov

Objectives

The goal of this project is to identify the barriers and approaches to achieving high-temperature coolant operation for the traction drive components of a hybrid vehicle and to identify the cost benefits/penalties when moving from a dedicated cooling loop at 65°C to the use of 105°C coolant from the engine radiator.

This assessment is important because automotive manufacturers are interested in utilizing the existing water/glycol engine cooling loop to cool hybrid electric vehicle (HEV) subassemblies to eliminate an additional coolant loop with its associated reliability, space, and cost requirements. In addition, the cooling of power electronic devices, traction motors, and generators is critical in meeting the FreedomCAR and Vehicle Technology (FCVT) goals for power rating, volume, weight, efficiency, reliability, and cost.

System tradeoffs will be taken into account in studying the use of high-temperature coolants. Corresponding radiator size, electronics packaging issues, motor housing size, pumps, hoses, power electronics, reliability of components, and meeting the 15-year FreedomCAR lifetime goal will be assessed against cost. The result of this effort will be a definitive analysis of the issues and tradeoffs involved with increasing the coolant temperature so as to ascertain the feasibility of moving ahead with 105°C coolant in hybrid vehicles.

Approach

The approach in this project is to review available literature and data to identify the barriers and approaches to achieving high-temperature coolants. A 50/50 water/glycol mixture will be used as a baseline coolant at 105°C in this investigation. Information will be collected from technical literature and industry.

Because heat flux is proportional to the temperature difference between the device's hot surface and the coolant, a device that can tolerate higher temperatures enables the device to be smaller while dissipating the same amount of heat. Trench insulated gate bipolar transistors (IGBTs) with higher-junction-temperature (175°C) silicon dies are gradually emerging into the market. This is an option that could provide more cooling margin in using 105°C coolant. Additionally, the permissible ripple current of various types of dc bus capacitors decreases rapidly as temperatures increase. New film dc-bus capacitors with higher-temperature capabilities are also emerging in the market. Necessary modifications to the existing cooling loop will be studied; cost, size, volume and weight tradeoffs assessed; and the results of the study reported.

The cooling of the power electronic devices, traction motors, and generators is critical in meeting the FreedomCAR goals for volume, weight, efficiency, reliability, and cost. Currently, the most commonly used coolant in automotive applications is a 50/50 water/glycol liquid mixture. The combustion engine is normally cooled by $<100^{\circ}\text{C}$ to reach 105°C or higher. Commercially available automotive-grade silicon electronic components have maximum junction temperature limits of 125°C . Therefore, the temperature difference between the junction and the 50/50 mix coolant of these devices is 20°C (i.e., $125^{\circ}\text{C} - 105^{\circ}\text{C}$). The less temperature difference that exists between the coolant and the dies' permissible junction temperature, the less margin there is to dissipate heat to the coolant environment.

This investigation focuses on identifying the barriers and approaches to achieving high-temperature coolants by analyzing the benefits and tradeoffs to using high-temperature coolants from the component, module, and system level perspectives.

Major Accomplishments

This study was performed by Oak Ridge National Laboratory (ORNL) to identify practical approaches, technical barriers, and cost impacts to achieving high-temperature coolant operation for certain traction drive subassemblies and components of HEVs. HEVs are unique in their need for the cooling of certain dedicated-traction drive subassemblies/components that include the electric motor(s), generators(s), inverter, dc converter (where applicable), and dc link capacitors. The new coolant system under study would abandon the dedicated 65°C coolant loop, such as used in the Prius, and instead rely on a 105°C engine cooling loop.

Because there is high interest by original equipment manufacturers (OEMs) in reducing manufacturing cost to enhance their competitive standing, the approach taken in this analysis was designed to be a positive, "can-do" approach that would be most successful in demonstrating the potential or opportunity of relying entirely on a high-temperature coolant system. Nevertheless, it proved to be clearly evident that a few formidable technical and cost barriers exist, and no effective approach for mitigating the barriers was evident in the near term.

The goals of the project were fully met by the identification of all significant barriers that pertain to the inverter, dc link capacitor, and motor. Approaches for resolving many of the barriers were also discussed; however, the full range of solutions will only come from additional R&D.

For operation at a significantly higher coolant temperature, component-level issues had to be addressed in this study. These issues generally pertained to the cost and reliability of existing or near-term components that would be suitable for use with the 105°C coolant. The assessed components include power electronic devices/modules such as diodes and IGBTs, inverter-grade high-temperature capacitors, permanent magnets (PMs), and motor-grade wire insulation. The need for potentially modifying/resizing subassemblies, such as inverters, motors, and heat exchangers, was also addressed in the study.

To obtain pertinent information to assist ORNL researchers to address the thermal issues at the component, module, subassembly, and system levels, preexisting laboratory test data obtained at varying temperatures were analyzed in conjunction with information obtained from technical literature searches and industry sources.

Technical Discussion

This project considers the potential temperature-related impacts on the performance of materials, components, and subsystems used in a Prius-like HEV traction system based on increasing the cooling loop temperature from 65°C to 105°C . The sections below focus on (1) the inverter power electronic devices, (2) inverter dc link capacitors, and (3) selected PM synchronous motor (PMSM) components that have the highest vulnerability to operation at elevated temperatures. The barriers that will be identified in each of these three main subsections will be those barriers that may prevent the components and subsystems from operating reliably during a 15-year service life when cooled by a 105°C cooling loop.

Note: The potential need for resizing the radiator was assessed, and it was determined that (1) the HEV heat load is small compared with that of the engine and (2) the air-to-coolant differential

temperature would be much higher, improving heat dissipation. Therefore, the need for a size change is considered minor or nonexistent.

Inverter/Converter Power Electronics Circuitry

This section considers what barriers may exist to operating the inverter and converter packages at 105°C. The analysis will consider inverter bus voltages of up to 500 Vdc for meeting peak power operation. Similarly, the Prius uses a 500-V boost converter, and the short-duration peak current in each leg of the inverter is $\sim 200\text{ A}_{\text{RMS}}$.

Conventional silicon-based materials and power device packaging designs are clearly challenging barriers to the use of a high-temperature coolant because junction temperatures will need to increase in high-temperature-coolant-based inverters to $\sim 150^\circ\text{C}$. For this reason, currently marketed but not fully life-tested devices will be considered in this analysis: silicon-based trench IGBTs and high-temperature diodes, both with junction temperature ratings of up to 175°C . These IGBTs and diodes are presently available from several manufacturers.

Although device specifications list “ 175°C ” maximum junction temperatures, the technology for these trench IGBTs and diodes is not mature; consequently, many design details must be considered that would normally have less importance. These include the determination of ratings for the actual parallel-device module packaging, observing current-dependent thermal deratings, observing temperature ratings for switching vs conductance, and the need to consider exactly how the module will be cooled in the final design.

On a positive note, Semikron’s recent efforts at developing a 133-kW high-temperature inverter system show the high potential of using even presently available power electronic devices. Their analysis showed¹ that the 40°C coolant rise did not require an increase in inverter size or in the number of power electronic devices. This was based on (1) using new high-temperature devices, (2) using devices with lower losses, (3) increasing the power density, and (4) improving the heat sink. Only a modest cost increase was projected for the power electronics.

This assessment of high-temperature power electronics did not identify any serious barriers, except for the use of conventional silicon-based power electronics. Instead, a listing was developed of what might best be characterized as weak barriers for the cooling of the inverter and converter subsystems at 105°C . This is based on individual assessments of the thermal performance of trench IGBTs and the potential for thermal fatigue and wire bonding failures in the power electronics device packaging.

The following describes both *potential* barriers for the inverter/converter and certain design considerations that merit mentioning:

1. As expected, conventional silicon-based materials and power device packaging cannot support the use of a high-temperature coolant because junction temperatures will need to increase to $\sim 150^\circ\text{C}$.
2. Recent literature² indicates that new 600-V IGBTs, using a trench metal oxide semiconductor top cell with ultra-thin 70- μm wafer technology, may be qualified for 175°C junction temperature, but the maximum operating temperature under switching conditions is reduced to 150°C . Because 150°C is much better than 125°C and most likely adequate based on development efforts, the thermal derating issue is a design consideration. More recently and subsequent to this study, Infineon Technologies announced 200°C junction temperature trench IGBTs.
3. There appears to be a significant potential for a solder-related fatigue issue that would limit the lifetime of IGBTs below 15 years if the use of a high-temperature coolant is implemented. This must be further investigated so that appropriate high-temperature solders, sintering, or other known technologies are employed.
4. The cooling requirements of the inverter driver board must be thoroughly evaluated and/or certain driver circuit components may have to be upgraded to maintain reliability/expected life at elevated temperatures.

Thermal Assessment of High-Temperature dc Link Capacitors

The capacitors used at the high-voltage dc-link-to-inverter interface are of interest because, within the bounds of current technology, it is very challenging to manufacture capacitors that provide adequate ripple current at the elevated temperatures. Even in a 65°C system, the present technology can barely meet this high-temperature application with acceptable volume, mass, cost, and reliability/lifetime. Highlighting this fact, the Prius capacitor module has a mass of 3.1 kg (almost 7 lb). In the case of the Prius, the modular polymer film capacitor takes up a significant portion of the volume inside the inverter/converter housing, as indicated in Figure 1.

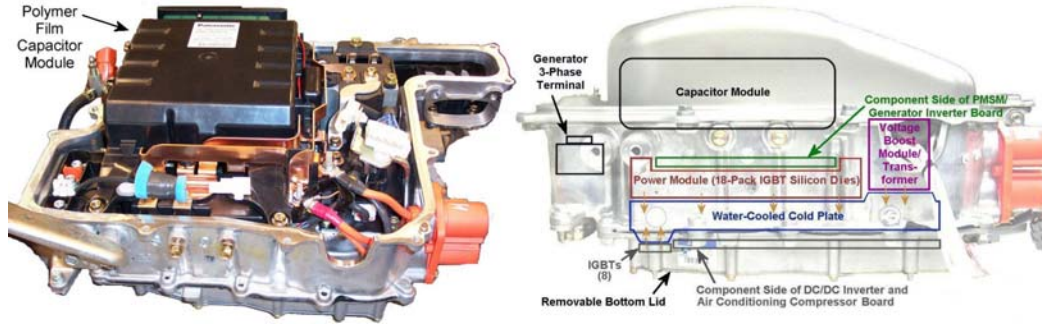


Figure 1. Modular capacitor position and size in the Prius inverter/converter casing.

The above concerns are certainly less of an issue for inverters placed in the much cooler passenger compartment as Honda does in its HEV line; however, that is outside the scope of this assessment, which considers only the use of a 105°C coolant for cooling of the HEV components and subsystems.

An engine compartment temperature of 140°C is considered applicable in the inverter design where a coolant-filled “cold plate” covers the base of a fully enclosed aluminum case in which the dc link capacitor is housed. Adopting a 105°C inverter coolant loop will increase the temperature of all the components in the casing, including the capacitor. There is also the potential for active capacitor cooling in some future HEV design.

Although many types of dielectrics exist, polymer film capacitors are generally produced with polypropylene, polyester, polycarbonate, or polystyrene. The temperature effects of these dielectrics result in changes in the capacitance, dissipation factor, and lifetime of film capacitors. High temperatures limit the available ripple current levels. Alterations in capacitance and dissipation factor play a large role in the amount of power dissipated and ultimately in the efficiency of the capacitor.

The amount of power dissipated across a capacitor is given as

$$P = V_{AC}^2 * 2\pi f * C * DF = \frac{I_{AC}^2}{2\pi f * C} * DF \quad (1)$$

The maximum heat removal capacity is

$$P = \frac{\Delta T}{R_{th}} \quad (2)$$

Solving these two equations for the current ripple yield

$$I_{AC} = \sqrt{\frac{(T_R - T_A) * 2\pi f * C}{R_{th} * DF}} \quad (3)$$

Assuming that a 500-μF capacitor has a case thermal resistance, R_{th} , of 130W/°C and that the capacitor is operated at a 5-kHz inverter switching frequency at rated temperature, the plots in Figure 2 show the maximum available ripple current of the capacitor vs temperature for polymer film and ceramic capacitors.

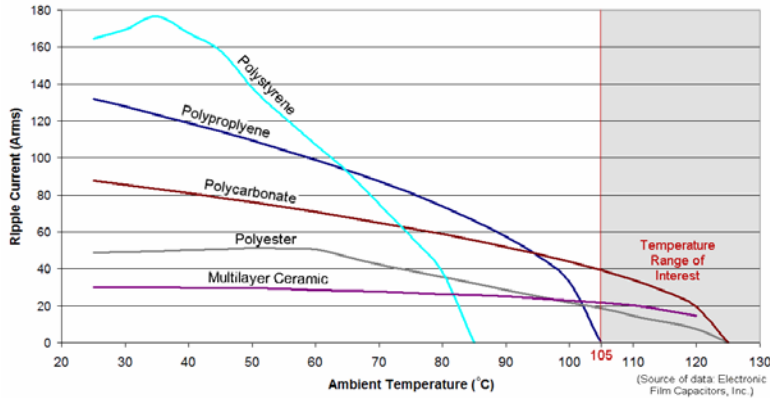


Figure 2. Ripple current vs ambient temperature for high-temperature capacitors.

Clearly, the adverse affect of temperature on ripple current is a serious issue. Even the polycarbonate, polyester, and ceramic capacitors do not appear to be of any potential use within the shaded range of interest. This leads to the question of what capacitors are being developed in laboratories or in the process of entering into production. Unfortunately, products in development are shrouded in secrecy, and details are considered to be proprietary. This has provided a serious obstacle to the assessment of high-temperature capacitors.

ORNL is able to report that a leading ceramic capacitor manufacturer produces a high-temperature capacitor for which the specifications are reportedly: 16 μF per module, 400-V continuous, $I_{ripple} = 25A @ 125^{\circ}C$ maximum, and the cost is \$20/module. Based on the required ripple current in an inverter, ~7 modules would be needed in parallel at a total cost of \$140 in large-quantity purchases. However, the 400-V limit is problematic because a 600-V rating for an appropriate margin in HEV systems includes a 500-V boost converter.

Other, higher-cost 400-V capacitors were reported, but there were no reports of newly developed 600-V, high-temperature, inverter-grade capacitors being available other than prohibitively expensive Teflon capacitors. However, Semikron³ reports that in the process of their development of a high-temperature inverter, they reached the conclusion that commonly available, 600-V capacitors can be used if three are placed in parallel. Using this approach instead of utilizing a high-temperature capacitor increases the volume, mass, and cost by a factor of 3; this is considered a serious penalty.

New capacitors with improved specifications and significantly lower cost are expected in 1 to 2 years—or in 10 years—depending on which technical or industry source is consulted.

The design barriers for dc-link capacitors include the following:

1. Temperature increases in the dc-link capacitor application have a strong and detrimental effect on the ripple current specification for many types of polymer capacitors.

2. High-temperature (125°C or higher) capacitors with adequate ripple current and a 600-V continuous rating can be found today by taking existing capacitors and using three in parallel. This approach triples cost and size, which is a serious penalty and a major barrier. There is a need for low-cost, high-temperature capacitors, and it is important that these be brought to the market soon.
3. The reliability and lifetime of any newly released, high-temperature capacitors must be well established through testing before they are introduced into vehicles.
4. Ceramic capacitors have superior high-temperature performance compared with polymer film capacitors; however, presently the cost is high, and an unacceptable energetic and flammable failure mode exists.
5. Teflon capacitors have excellent high-temperature performance, but cost is clearly prohibitive.

Thermal Characterization of Magnets

Neodymium-iron-boron (NdFeB) magnets are the most powerful of all the magnets commonly used in industry and are made of advanced rare-earth materials. They provide high performance and resilience against demagnetization. However, these magnets are extremely sensitive to temperature and susceptible to oxidation if not properly coated.^{4,5}

NdFeB magnets are frequently selected for HEV applications, including the Prius, because of their high magnetic strength and acceptable cost. Unique Mobility (UQM), a major manufacturer and researcher of PM motors, also uses NdFeB PMs.⁶⁻¹⁰

An increase in temperature of NdFeB PMs reduces the magnetic strength linearly up to 100°C as follows:¹¹

$$B_r = B_{rA} + s(T - T_A) ,$$

where B_{rA} is the residual flux density at normal ambient temperature T_A , and s is the slope of the B_r temperature characteristic (for NdFeB this tends to be -0.1% per °C). Above 100°C, the magnetic field strength begins to fall more rapidly as indicated in Table 1.^{7,12}

Table 1. Characteristic of Delco Remy MQ2^a NdFeB magnets

Residual induction	0.8T
Intrinsic coercive force	1.43 ma/m
Temperature coefficient of bromine to 100°C	-0.10%
Temperature coefficient of bromine to 125°C	-0.11%
Temperature coefficient of bromine to 175°C	-0.15%

^aMQ2—Delco Remy Magnequench Division

For an NdFeB PM with the above characteristics, Figure 3 shows the reduced strength of the magnet in per unit form vs temperature. It is estimated that there will be a 20–30% reduction in flux strength for motor operation at the higher coolant temperature as PM temperature approaches 200°C. This temperature-induced effect represents a very significant barrier. This reduction in magnetic field strength due to temperature is reversible because the magnetization will return to the original value once the temperature is decreased.

If the operating temperature of the motor exceeds a certain critical temperature, irreversible demagnetization of the magnet will result.⁹ This critical temperature is $\sim 150^\circ\text{C}$ for many types of NdFeB magnets, but 220°C for the particular type of NdFeB PM selected for the Prius.¹³ However, the reversible, temperature-induced reduction in field strength is more significant because the motor performance (i.e., shaft power) will be significantly reduced at temperatures that will be frequently reached during vehicle operation (assuming a 105°C coolant).

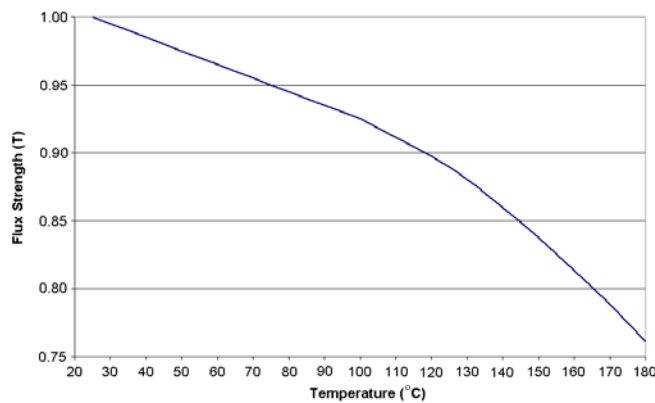


Figure 3. Flux strength of a selected NdFeB magnet vs temperature.

Stator Winding Insulation

Magnet wire used in many applications today is inverter spike resistant (ISR). This Class H wire has excellent resistance to damage from high-voltage spikes that commonly result from rapidly changing pulse-width modulated (PWM) currents in inductive circuits.

Motor wire insulation is a well-developed technology that is not expected to change significantly in the next several years. That is not to imply that major strides have not been made in the last 25 years. Today's motors operate at significantly higher temperatures while meeting 20,000-h lives or greater. The most common types of insulation, in order of increasing quality, are Class A, Class B, Class F, and Class H. These insulations are rated for maximum temperature in 25°C increments as indicated in Figure 4. Less common insulations are available such as Class S (240°C) and Class C (over 240°C); however, the use of these would create a cost increase, which may be avoidable.

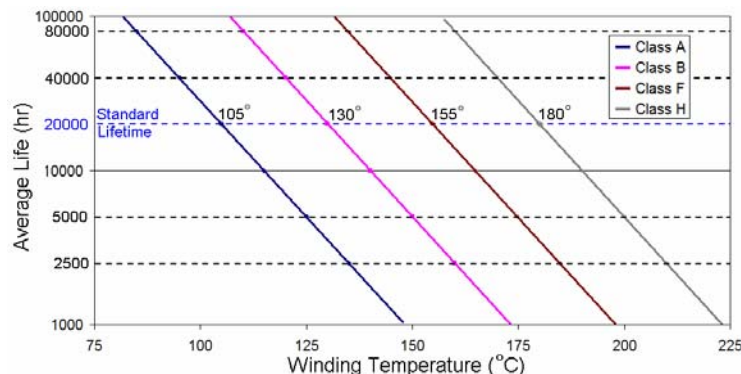


Figure 4. Average motor insulation life vs winding temperature for four insulation types.

For HEV applications, it is essential to fully understand the duty cycle and rate of accrued time on the motor when estimating motor insulation life. For instance, the Prius is considered to be a "strong hybrid" since the motor is relied upon 100% for accelerations from a full stop. It is also used to assist the engine for accelerations and high torque demands such as passing vehicles at highway speeds and for driving up a grade. Table 2 shows an estimate of the required number of hours that may be expected from the motor during a 15-year period. The example numbers reflect heavy usage of the vehicle with an accrued

Table 2. Estimate of life-shortening hours accrued during 15 years of city driving

Daily vehicle operation (h/day)	Days in a year	Number of years	Fraction of vehicle-operation time when the stator insulation is at a life-shortening temperature	Accrued life-shortening hours in 15 years	Accrued miles in the 15 years @ 35 mph
1.25	365	15	0.75	5,100	240,000

240,000 miles. If one assumes the fraction of time when the motor temperature is hot (near or above 180°C) to be 0.75, then the number of life-shortening hours on the insulation is 5,100 h. This high fraction would reflect stop-and-go city driving rather than highway usage where the motor may be seldom used. The 5,100 h is far below the 20,000-h standard for motor insulation life. Referring to Figure 4, this would necessitate the use of Class H insulation at 200°C instead of the 180°C standard.

PMSM Barriers

Before identifying design barriers, the following observations are made concerning an unmodified PMSM that is cooled with a 105°C cooling loop:

1. Heat will not be removed from the casing as effectively as in the case of a 65°C coolant design.
2. Because of (1), the oil temperature, stator temperature, and rotor temperatures will be higher.
3. There will be significant time periods when the 105°C coolant will be *heating* the motor casing.
4. There will also be times when the 105°C coolant will provide *cooling* due to (a) the temperature floor of the PMSM being raised and (b) motor operation in engine compartment temperatures of up to 140°C.

Figure 5 shows the general design approach for a PMSM cooled by 105°C coolant. Because the temperature floor is raised substantially, a design response is needed to lower and stiffen the temperature ceiling by making the heat exchanger, and possibly other components, more effective in responding to thermal excursions that occur during vehicle operation.

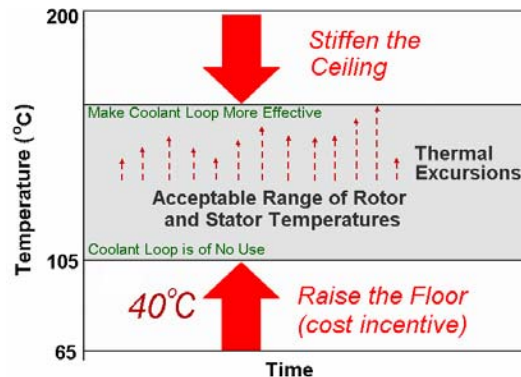


Figure 5. General PMSM design approach applicable for high-temperature cooling.

The following are the PMSM barriers:

1. The primary barrier for the PMSM is that the flux strength of the PMs will decrease 20–30% whenever the PMs are at elevated temperatures consistent with operation based on a high-temperature coolant. This will reduce PMSM/vehicle performance and/or require higher, compensating motor current. Higher motor current will compound the thermal control problem.

2. As determined in the electrical assessment, higher temperatures result in a very substantial 29% increase in losses due to temperature-induced changes in back-emf and stator winding copper losses. This will result in unacceptable temperatures for the stator insulation and PMs unless the cooling system is significantly improved and/or the size of the motor is increased to facilitate heat transfer. Detailed thermal studies will be required to produce a high-temperature motor design.
3. During those times when heat flows from the motor to the coolant, the temperature gradient between the coolant and casing will be low, reducing the effectiveness of the heat exchanger.
4. Because of increased stator winding operating temperatures, Class H insulation will be operated beyond its conventional limits. This may be acceptable based on total hours of operation at high temperatures, but some level of detailed verification/study may be needed.
5. Higher-temperature stator insulation (Class C) can be used but at higher cost.
6. One design approach to better controlling PM temperature entails the design of a more effective water/glycol-coolant-to-oil heat exchanger. Increasing the size of the heat exchanger and locating it at the bottom of the PMSM are desirable approaches but will increase volume, mass, and cost. Encircling the stator with a cooling jacket is another option. These approaches are considered to be “a beginning” that must be followed by other, yet-undefined improvements.
7. The acceptability of raising coolant temperature can only be fully verified by selecting detailed candidate designs and performing thermal analysis and performance modeling. This will involve developmental costs.

System-Level Assessment

In considering numerous design changes that must be or could be pursued to eliminate a dedicated, low-temperature cooling loop, it is important to weigh the cost requirements of each proposed design against the cost savings of eliminating the dedicated cooling loop.

The Prius uses a \$480 radiator (2.05 g and 4.19 L) that is partitioned to provide a low-temperature, 1.14-L, dedicated coolant loop radiator. This loop also uses a \$142 12-Vdc electric pump (344 g) and miscellaneous hoses. The elimination of this system may entail simply eliminating the partitioning in the radiator, eliminating the pump, and reconfiguring a few hoses. Table 3 shows the predicted cost savings. Using an unpartitioned radiator would save ~\$80 and, with the elimination of the pump and two hoses, the total savings would be ~\$250, based on the retail prices of new components. The wholesale cost would be ~\$187.50.

Table 3. Itemized listing of costs of eliminated components

Eliminated components or features	Cost savings (\$)	Source used as basis of estimation
Electric pump	142	Toyota of Knoxville
Partitioning of the radiator	80	—
Hoses (2) associated with eliminated components (pump and partitioned radiator)	28	toyotapartscheap.com
Total retail	250	
Minus 25% retail store profit	-62.5	
Wholesale	187.50	

Conclusion

This study consisted of a technical/engineering evaluation and literature search designed to investigate the feasibility of raising the coolant temperature of a Prius-like HEV system from 65°C to 105°C. The study focused on subsystems and components that were deemed most vulnerable to operation at higher temperatures. These included the inverter power electronics, the dc link capacitor for the inverter, and stator insulation and PMs in the PMSM.

There is high interest among the OEMs in reducing manufacturing cost to enhance their competitive standing. One candidate means of accomplishing this is eliminating the HEV-dedicated coolant loop. In

light of this high interest, a positive, “can-do” approach was used in this study to the fullest extent possible to fairly assess the potential or opportunity of relying entirely on a high-temperature coolant system. Nevertheless, it proved to be clearly evident that a few formidable technical and cost barriers exist, and no effective approach for mitigating the barriers was identifiable in the near term.

The major barriers encountered in this study are

1. **Inverter dc link**—High-temperature (125°C or higher) capacitors with adequate ripple current and a 600-V continuous rating can be found today only by taking existing capacitors and using three in parallel or by using prohibitively expensive Teflon film capacitors. The first approach triples cost and size, which is a serious penalty and a major barrier. There is a need for low-cost, high-temperature capacitors, and it is important that these be brought to the market soon.
2. **PMSM1**—The primary barrier for the PMSM is that the flux strength of the PMs will decrease by 20–30% whenever the PMs are at elevated temperatures consistent with operation based on a high-temperature coolant. This will reduce PMSM/vehicle performance and/or require higher, compensating motor current. Higher motor current will compound the thermal control problem.
3. **PMSM2**—Higher temperatures result in a very substantial 29% increase in losses due to temperature-induced changes in back-emf and stator winding copper losses. This will result in unacceptable temperatures for the stator insulation and PMs unless the cooling system is significantly improved and/or the size of the motor is increased to facilitate heat transfer. Detailed thermal studies will be required to produce a high-temperature motor design.

Although there were no “major barriers” for the inverter/converter power electronics, one issue remains. New 600-V IGBTs, using a trench MOS top cell with ultra-thin 70-μm wafer technology, may be qualified for 175°C junction temperature, but the maximum operating temperature under switching conditions is reduced to 150°C. Because 150°C is much better than 125°C and most likely adequate, the thermal derating issue is mainly a design consideration. Furthermore, very recently, Infineon Technologies released new devices that can reportedly operate with a 200°C junction temperature without the need for current derating.

Table 4 lists the potential cost increases that would result from resolving the barriers for the major critical subsystems/components identified in this assessment. Other costs may be incurred for such things as inverter driver circuit improvements, the buck/boost converter high-temperature capacitor, improved oil spray design for stator cooling, etc. However, the costs of the items listed in Table 4 already exceed the \$188 cost savings of eliminating the dedicated coolant loop by a substantial amount. Even the summation of the *minimum* estimated costs exceeds the cost savings.

Table 4. Estimations of additional costs to resolve barriers in components/subsystems

Subsystem or component	Constraints specified by the assessment	Estimates of additional cost
Inverter power electronics	The <i>future</i> cost increase for high-temperature power electronic modules when they become available for full rated switching current without thermal derating.	\$25–125/inverter \$4–21/converter
dc link capacitors	The cost increase of using three times the capacitance prior to new products becoming available with adequate ripple current and a 600-Vdc continuous rating.	\$210–250 ^a
PMSM (protecting PM and stator insulation)	The cost increase for additional casting in the heat exchanger and other as-yet unknown improvements necessary for control of PM temperatures and dissipating higher losses.	\$30–200
Total		\$269–596

^aThis assumes the cost of a low-temperature capacitor, similar to that used in the Prius, is ~\$100.

Table 5 is a qualitative comparison matrix of various parameters of interest for each of the critical components and subsystems. The parameters of interest include availability of components, mass, volume, performance, reliability, lifetime, and cost. For temperatures lower and higher than the 65°C base case, the potential thermal effect is designated as either being positive (+) or negative (–), where applicable. Major barriers are designated with double negatives. As an example using stator insulation, adopting a 105°C coolant would have negative impacts on mass, volume, and cost *if* it were necessary to use a different type of insulation, such as Class C. This assessment considered remaining with Class H to be potentially feasible, depending on whether heat transfer can be significantly improved, in which case those negatives would not apply. This illustrates how some designations in the table are only *potentially* applicable.

Table 5. Multiparameter, temperature-based qualitative comparison

Parameter	50°C	65°C	105°C
<i>Elimination of HEV-dedicated cooling system</i>			
Cost		Base case	+
<i>Inverter power electronics</i>			
Availability of components		Base case	
Mass		Base case	
Volume		Base case	
Vehicle performance		Base case	
Reliability	+	Base case	–
Attaining a 15-year life		Base case	–
Cost		Base case	
<i>dc link capacitor</i>			
Availability of components		Base case	– –
Mass		Base case	–
Volume		Base case	–
Vehicle performance		Base case	
Reliability	+	Base case	–
Attaining a 15-year life	+	Base case	–
Cost	+	Base case	– –
<i>PMSM PMs</i>			
Availability of components	+	Base case	–
Mass		Base case	
Volume		Base case	
Vehicle performance	+	Base case	– –
Reliability	+	Base case	–
Attaining a 15-year life	+	Base case	–
Lack of a new PMSM design ^a		Base case	– –
Cost	+	Base case	–
<i>PMSM stator insulation</i>			
Availability of components		Base case	
Mass		Base case	–
Volume		Base case	–
Vehicle performance		Base case	–
Reliability		Base case	
Attaining a 15-year life		Base case	–
Lack of a new PMSM design ^a		Base case	– –
Cost		Base case	–

^aA new design is needed for significantly improved heat dissipation.

Future Direction

The high-temperature barrier study is complete as of the end of FY 2006. The study served to highlight a number of areas needing attention, and there are potential areas in which the FCVT program may see a new (or renewed) need to support development. These areas include the development of low-cost, high-temperature components and packaging concepts that meet all the required specifications. The components include IGBTs, diodes, inverter-grade capacitors, and PMs. Innovations in cooling systems, such as oil-spray cooling of the rotor/stator, are also needed for the development of a viable high-temperature PMSM.

Publications

R. H. Staunton, J. S. Hsu, and M. R. Starke, *Barriers to the Application of High-Temperature Coolants in Hybrid Electric Vehicles*, ORNL/TM-2006-514, Oak Ridge National Laboratory, 2006.

References

1. Communication from J. Mookken, Semikron, to R. H. Staunton (ORNL) via email, *Inverters for High Temperature*, and phone on August 7, 2006.
2. P. Kanschat, H. Ruthing, et al., "600V-IGBT: A Detailed Analysis of Outstanding Static and Dynamic Properties," presentation at PCIM Europe Conference, 2004.
3. John Mookken, Semikron, "High-Temperature (105°C) Inverter Development," presentation at DOE FreedomCAR Vehicle Technology Program Annual Review, Oak Ridge, Tennessee, August 2006.
4. R. H. Staunton, S. C. Nelson, P. J. Otaduy et al., *PM Motor Parametric Design Analyses for a Hybrid Electric Vehicle Traction Drive Application*, ORNL/TM-2004/217, Oak Ridge National Laboratory, Oak Ridge, Tennessee, September 2004.
5. Anil Naji, "High Performance Permanent Magnets 7," Catalog, Magnet Sales and Manufacturing Inc, 1995.
6. Arnold Technologies, "TechNotes: How to Select the Appropriate Permanent Magnet Material," December 2002.
7. J. G. West, "Propulsion Systems for Hybrid Electric Vehicles," *Electrical Machine Design for All-Electric and Hybrid-Electric Vehicles (Ref. No. 1999/196)*, IEE Colloquium, pp.1/1–1/9, 1999.
8. *Annual Report 2005*, UQM Technologies, Frederick, Colorado, 2005.
9. T. Sebastian "Temperature Effects on Torque Production and Efficiency of PM Motors Using NdFeB Magnets," *IEEE Transactions on Industrial Applications*, **31**(2), April 1995.
10. F. Crescimbeni, A. D. Napoli, L. Solero, and F. Caricchi, "Compact Permanent-Magnet Generator for Hybrid Vehicle Applications," *IEEE Transactions on Industrial Applications*, **41**(5), October 2005.
11. Akira Kawahashi, "A New Generation Hybrid Electric Vehicle and Its Supporting Power Semiconductor Devices," Toyota Motor Corporation, *Proceedings of 2004 International Symposium on Power Semiconductor Devices and ICs*, 2004.
12. J. F Gieras and M. Wing, *Permanent Magnet Motor Technology*, Marcel Dekker, 2002.
13. Verbal communication from Dr. Hitoshi Yamamoto, Neomax America, Inc., to J. S. Hsu, ORNL, during the 2005 Permanent Magnet Conference, 2005.

2.3 Direct Cooling Options: An Assessment of Air Cooling for Use with Automotive Power Electronics Applications

Principal Investigator: Desikan Bharathan

National Renewable Energy Laboratory

1617 Cole Blvd.

Golden, CO 80401 USA

Voice: 303-887-4215; Fax: 312-235-3703; E-mail: Desikan_bharathan@nrel.gov

DOE Technology Development Manager: Susan A. Rogers

Voice: 202-586-8997; Fax: 202-586-1600; E-mail: Susan.Rogers@ee.doe.gov

Abstract

Air cooling of electronic components using high-speed airflow that is in direct contact with hot electronic components is assessed. Strict requirements for power conversion devices as needed for automotive applications are addressed. Continuing work in air cooling in FY07 is outlined. Recommendations for future course of research in this area for thermal control of power electronics are suggested.

Objectives

The overall effort in this area falls under the general program for thermal control of the Advanced Power Electronics and Electrical Machines (APEEM) program. The National Renewable Energy Laboratory (NREL) leads research and development activities in thermal control related in the APEEM program. We provide an overview of the activities that fall under the thermal control activities.

The overall objective of the thermal control activities is to develop advanced technologies and effective integrated thermal control systems, aimed to meet U.S. Department of Energy (DOE) FreedomCAR program goals. The FreedomCAR goals address key requirements for power electronics, such as target values for volumes, cost, and weight of various subcomponents. The barriers for thermal control technologies are affected by the overall specifications related to these quantities. These limitations are:

- Volume and temperature limits. Existing systems are bulky and difficult to accommodate for automotive applications. Existing thermal control techniques are inadequate to dissipate high-heat fluxes (~ 250 W/cm²) while limiting the operation of silicon-based electronic components to a temperature of less than 125°C.
- Cost. Material and processing technologies remain costly for use in automotive industry, which is highly cost competitive.
- Weight. Current components are generally considered heavy resulting in needs for additional structural support and increased use of parasitic power.

To achieve the specific FreedomCAR goals for 2010, significant advancement in thermal control of both the power electronics and motors for electric propulsion system must be achieved. Through optimization of existing technologies and their extension to new pioneering cooling methods, we intend to achieve higher power densities, smaller volumes, weights, and increased reliabilities for the drivetrain components. Detailed investigations into thermal issues can provide viable paths to bridge gaps to achieve of FreedomCAR technical targets. It is intended that these paths will also lead to lowering the costs for such new technologies to allow their applications in the automotive industry.

The thermal control and management of electronics, and electronic systems represent major technical barriers to achieving specific FreedomCAR goals in 2010. Electronic chips used as switches in power conversion devices operate at high currents and extreme heat loads resulting in hot spots. Thermal performance of a power module containing such switches may be measured by the maximum temperature rise in the switching die at a given power dissipation level with a fixed heat-sink temperature. The lower the die temperature, the better its electric performance and reliability will be. As the thermal resistance from junction of the die to the heat sink is reduced, higher power densities can be achieved for the same temperature rise, or for the same power density, lower junction temperature can be attained. It is also important to reduce thermal excursions to improve the life and reliability of the dies.

The problem areas of concern in thermal control of power electronics are many including: operating temperature of insulated gate bi-polar transistors (IGBTs); thermal contact resistance between various layers of a power module; low thermal conductivity of the thermal paste; maximum heat flux that can be removed (ideally faster IGBTs would reject heat at a rate of about 250 W/cm^2); limitations on the inlet cooling fluid temperature (it is desirable to use the ICE coolant at 105°C); cost of the cooling technology in terms of its complexity, weight, and volume.

The existing cooling technologies are depicted in Figure 1. This figure shows that conventional cooling techniques, such as forced convection and simple 2-phase boiling techniques, are not capable of removing target level of heat fluxes (in the range of 250 W/cm^2) at low temperature differences (20°C). However, this figure shows clearly that enabling technologies—such as spray cooling and jet impingement, along with innovative packaging improvements—can help meet the goals of the FreedomCAR program. It is, therefore, our objective to investigate and establish, through modeling and testing, the potentials for these enabling technologies to achieve the desired goals.

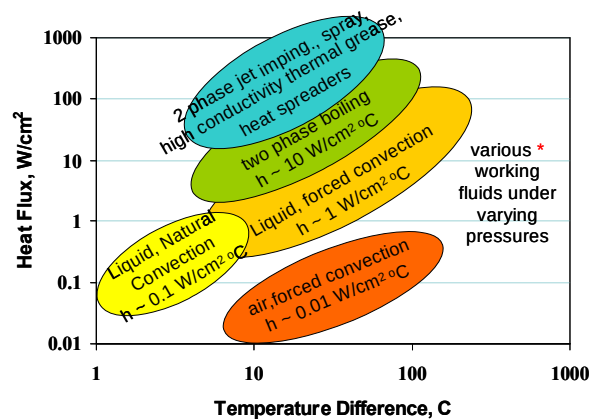


Figure 1: Capabilities of existing cooling technologies

Ideally, it would be more beneficial if the IGBTs could be designed to operate at higher temperatures. Through use of silicon carbide and other innovative materials, DOE's APEEM program is also pursuing parallel, longer-term research projects to evaluate and achieve that objective. However, to meet today's needs of the U.S. industry, the existing silicon-based IGBTs should be operated at temperatures below 125°C .

Existing inverter power modules are constructed by bonding die, copper layers, and a substrate together; and these layers then are mounted on a base plate with an intervening thermal grease layer to

accommodate thermal stresses. Figure 2 shows the structure of the mounted die on a cold plate with the heat being removed by a coolant supplied to one side of the die. Various layers of the construction are indicated. A major contributor to the thermal resistance is the thermal grease. While thermal interface materials are used to promote heat transfer between the DBC and cold plate by filling in voids with conductive material, the thermal interface material layer remains a significant thermal resistance. In addition, the thermal grease is used to reduce thermal stress that may result from differential thermal expansions amongst the various layers. These greases can withstand high temperatures, but have very low thermal conductivities of about 0.3 W/mK to 0.5 W/mK. As a result of this low thermal conductivity, the thermal grease constitutes about 30% to 40% of the total thermal resistance between the silicon die and the heat sink. It is also crucial to reduce or even eliminate this resistance by increasing the thermal conductivity of the thermal grease.

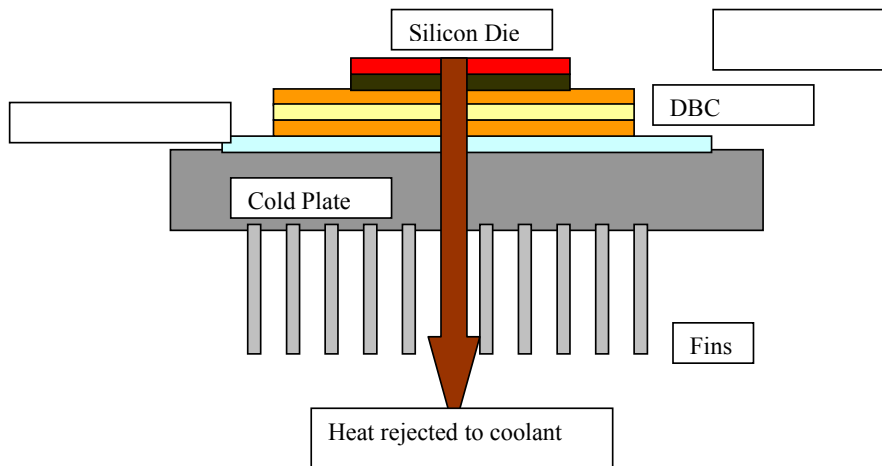


Figure 2. Schematic diagram of the cooling path for removing heat from a silicon die

Another approach for removing higher heat fluxes from the IGBTs is to spread out their heat flux over a larger surface. Currently, copper or aluminum base plates spread the heat over an increased area of about 20% to 30%, thus reducing the maximum heat flux at the coolant-to-heat-transfer-surface interface. Use of more effective heat spreaders—such as highly conductive layers, mini-heat pipes, and/or phase change materials embedded within the cold plate—can spread out the heat flux over larger areas and reduce the maximum heat flux by as much as 40%. With such a reduction in the maximum heat flux, the removal rate may be met by one of the enabling cooling technologies, such as jet impingement and spray cooling.

Another approach to maintaining the power module's temperature at 125°C or below is to provide a separate cooling loop where the coolant can enter the heat sink at lower temperatures, hence providing a larger temperature difference to maintain the IGBT's temperature at the desired value, while removing high-heat fluxes.

A last approach is the use of air on both sides of the silicon switches. Use of air for cooling has many advantages. Air is the ultimate heat sink for all heat rejection from an automobile. All the heat that is to be rejected, either directly or through the use of an intermediate coolant loop must end up in air. Direct use of air can eliminate many components of the cooling loop and the necessity for carrying a secondary coolant. Ambient air is assumed to be available at a nominal 30°C for all the vehicle cooling strategies.

Use of such air increases the overall temperature-driving potential available to reject the heat. Air, as the cooling medium, is benign, nontoxic, and free when compared to many other fluids. Airflow is also amenable to be modulated in a transient manner to suit the needs the system, as the system load varies during operation.

Despite the many advantages, air cooling has many drawbacks as well. Air remains a poor heat transfer medium, with low thermal conductivity and low density. A fan or a compressor must move the airstream to the required hot spots that result in additional parasitic power loads. The coefficient of performance for such a system may turn out to be low.

Cooling with air may not resolve problems related to the current generation of silicon-based devices. But as technology changes, the trench IGBTs can operate at 175°C, and the more advanced Si-C based devices are projected to operate at even higher temperatures. It is with these devices, that air cooling may become a practical option.

To address these thermal control issues, NREL has undertaken many distinct areas of research and development. Multiple avenues of research are expected to ultimately result in significant advancement in thermal control of power electronics to meet the target goals and objectives of the FreedomCAR initiative. The following describes progress made toward evaluating air cooling as a viable option for power electronic components in FY2006 under the FreedomCAR program.

Overall Objective

The objective of the present task is to assess the potential for reducing the cost and complexity for the cooling system for power electronics using air. This study and assessment are aimed to result in quantifying the relative merits for the use of air for cooling power electronic devices to achieve high-heat flux removal rates under steady-state and transient conditions. The cooling system could also be modulated to match both the base line and the transient cooling needs, and thus perhaps needs to be sized to handle only the peak transients for short periods of time. The parasitic load for running such a cooling system can be minimized by a proper approach to optimize the system weight and cost.

In conjunction with utilizing an air-cooling system, NREL evaluated a compressed air system as well. This system is similar in concept to the Brayton air cycle; it is run as a heat pump rather than a power cycle. A compressed air system might achieve a coefficient of performance of the order of 2 to 3. An air volume of 1 liter, compressed to a pressure of 5 bars, when expanded to atmospheric pressure, can reach temperatures as low as -50°C or lower. Such a pulse of air can remove a quantity of heat on the order of 40 to 80 J using pulses lasting 1 to 2s.

This project assessed air cooling of power electronics over a range of outside air temperatures; air velocities; degrees of mixing of outside air and engine compartment air; and other parameters. The goal is to determine under what conditions air cooling would be acceptable.

Use of compressed air for cooling power electronic systems that result in hot spots with high-heat fluxes warrants a closer look. The load on the power electronic systems is highly transient with large peaks occurring over very short durations (with peak to average ratio of 20 or larger, lasting over 1 to 3s time frame). Depending on how the electronic device may be constructed, the heat capacity of the device may be low to cause large temperature swings on the device over the duration of peak demand. Pulsed compressed air jets may be used to effectively remove the high-heat fluxes during such transients. During normal demands, a conventional system may carry the base load for heat removal.

To evaluate such a system using compressed air, we needed to generate a detailed load profile for the power electronic system for various standard driving cycles. The detailed transient model for the compressed air system may then be developed using standard thermodynamic principles. The system can be evaluated with models for transient heat transfer occurring over the hot spots. Once the sizes for various components can be arrived at, the overall system cost can be evaluated and assessed as to their viability for use. Use of a compressed air tank in the vehicle that can be refilled at a gas station offers an alternative approach to this evaluation. Infrastructure for such compressed air already exists. Temperature monitoring circuitry can be used to activate air pulses as and when necessary to minimize the airflow requirements for the system.

This effort was carried out as part of the thermal management tasks under the DOE APEEM program under the Office of Energy Efficiency and Renewable Energy.

Detailed Objective

The overall objective is to assess the suitability of air cooling for heat removal from power electronic components being developed as part of the FreedomCAR effort under DOE.

The overall mission of the FreedomCAR program addresses many components of the vehicle. Achieving the FreedomCAR goals will require the development of new technologies for power electronics and electric machinery. The new technologies must be compatible with high-volume manufacturing; ensure high reliability, efficiency, and ruggedness; and simultaneously reduce cost, weight, and volume. Key components for fuel cell and hybrid-electric vehicles include motors, inverters/converters, sensors, control systems, and other interface electronics.

One of the specific objectives related to power electronics is to develop automotive integrated modular power modules for power conversion and control. Building blocks to be developed include power switch stages optimized to run newly developed motors, drives, DC/DC converters, and advanced controllers and sensors.

The technical targets as specified by the Energy Efficiency Technology Team (EE Tech Team) are summarized below. The FreedomCAR goals and technical targets for 2010 for the electric-propulsion system, as given in Table 1, include the motor, inverter, gearbox, and controller.

Table 1. Electric Propulsion System Goals and Technical Targets for 2010

FreedomCAR Goals	
Peak power	55 kW for 18 seconds
Continuous power	30 kW
Lifetime	>15 years (150,000 miles)
Cost	<\$12/peak [<\$660] ^a
Technical Targets	
Specific power at peak load	>1.2 kW/kg [<46 kg] ^a

Volumetric power density	>3.5 kW/l [<16 l] ^a
Efficiency (10 to 100% speed, 20% rated torque)	90%

Numbers in square brackets are equivalents for a 55 kW peak-power system.

Of specific interest to this study is the inverter/converter where the electronic chips are expected to generate heat fluxes of up to 250 W per square centimeter of the silicon-based external chip contact surface. Current technology based on silicon chips requires that the chip maximum temperature not exceed 125°C to assure their reliable performance over the required life of 15 years. To reduce the complexity and perhaps the overall cost for the electronic cooling system, the technology team also set other system requirements such as: the coolant to be used will be the same water-ethylene glycol mixture that cools the internal combustion engine, and that this coolant will be available at a return temperature of 105°C.

These requirements, as set by the EE Tech Team, dictate the use of a particular fluid and limit the choices of methods that can be adopted to achieve high heat removal capabilities. However, parallel activities within the APEEM program generated additional tasks to look into technologies that might have the potential to yield substantially higher heat removal capabilities, and/or reduce the complexity of the cooling system, or both.

Use of air for heat removal falls under the category of technologies that could potentially meet the heat removal requirements with greater ease and lower cost.

Approach

Our overall approach to investigating air cooling as an option for power electronics included a review of literature, and an investigation of high-heat flux air cooling used in mission-critical applications, such as turbine blade cooling, and other similar applications that occur in hot-wire anemometry. We attempted to assess the hardware options that may be available in the industry for varied air-cooling devices. We conducted system level analyses to identify areas of critical needs. We also conducted computational analyses of fluid flow and heat transfer for promising devices and geometry. Based on the collective information of all these efforts we arrived at recommendations for future activities in this area.

Our attempt aims to enhance the three key parameters that affect the overall heat transfer rate, namely the air-side heat transfer coefficient, area that is in contact with air, and the overall temperature driving potential. The approach to enhancing the heat transfer coefficient is to attempt to establish high-speed airflow with thin boundary layers that may be interrupted often to capture an entrance-region type of flow. Use of pins and other types of fins is to be investigated for increasing the area of contact. Air refrigeration is also attempted to increase the temperature-driving potential. Details of the various approaches are summarized in the following sections.

Major Accomplishments

Initial assessment for air cooling focuses on a simple heat transfer that occurs on a single chip. We assume a chip frontal area of nominal 1 cm² with the chip generating a heat load of 200 W. Air at ambient pressure and a temperature of 30°C is used in arrangement with a jet schematically shown in Figure 3.

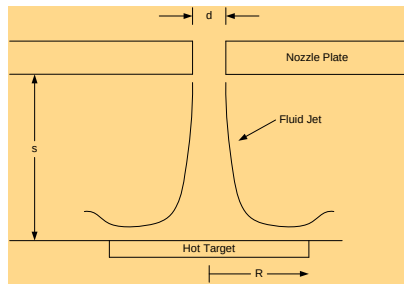


Figure 3. Schematic diagram of an air jet impinging on a hot surface

Use of Ambient Air with Conventional Devices

Figure 4 shows a flow sheet for cooling with air developed using the commercially available software ASPEN. The chip shown as a block is maintained at a temperature of 125°C (using steam for the purposes of the simulation).

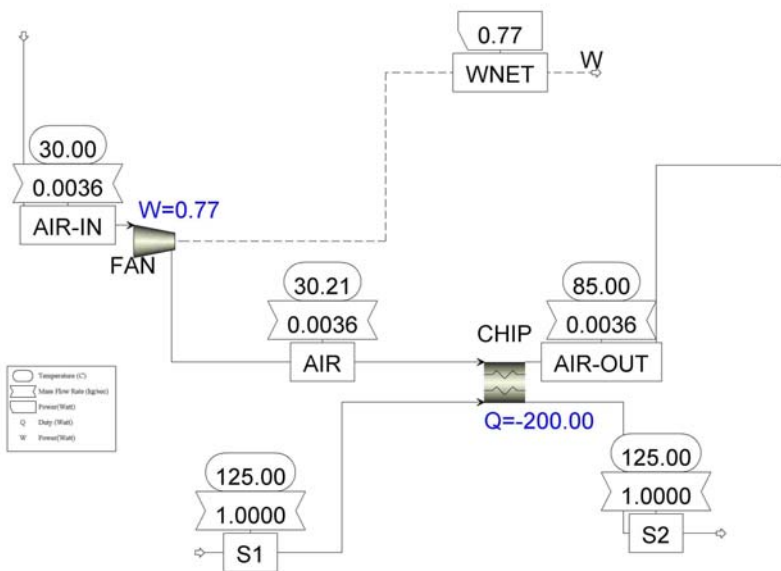


Figure 4. Flow sheet for air cooling with a target chip generating 200W heat load

Only key parameters are shown in this figure. The air approach temperature is fixed at 40°C below that of the target hot surface, a value typical for airflow applications. The fan operates with an efficiency of 56 %, once again typical for airflow.

Based on this simulation, we find that the airflow requirement per chip is about 3.6 g/s (or 2 ACFM), and the required (UA) product, that is the product of the heat-transfer coefficient and the available area for heat transfer, is 3.15 W/K. Typical industrial airflow applications nominally achieve a heat transfer coefficient of about 40 W/m²K. If we use this value, the required area for achieving the above stated conditions is about 800 fold. That is, the area must be enhanced by a factor of 800 through the use of fins or other means. We find that nominal heat transfer coefficient falls well below the needs to accomplish the task at hand.

Use of refrigerated air with conventional devices:

Compressed, cooled air can be expanded to yield even lower temperatures. Figure 5 shows a schematic arrangement for such a system. The general consensus is that such a refrigerated system is perhaps too complicated to be of use in automotive applications. The system consists of a compressor that is coupled with an expander and heat rejection system in between. Power from expansion is used in the compressor to offset the required work input. Nominal values for the efficiencies for the various components were used in evaluating this system, such as setting the turbine at 90%, the compressor at 80%, and all air-cooler approach temperatures at 40°C.

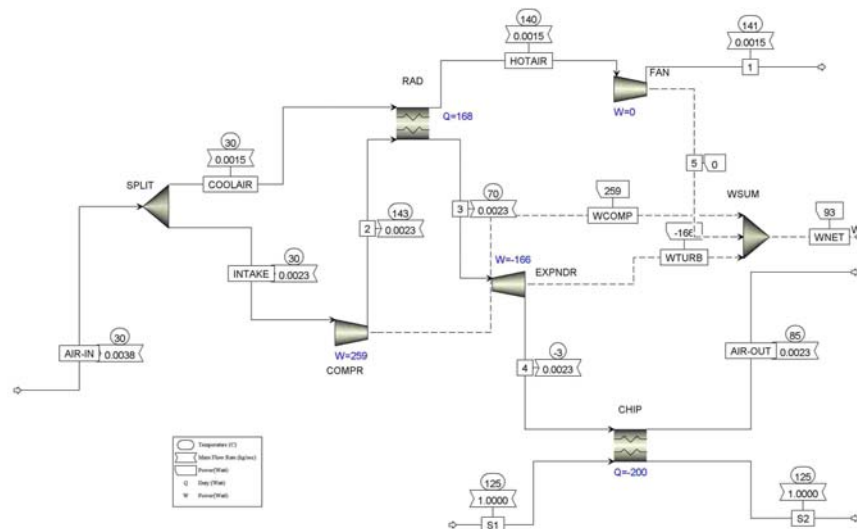


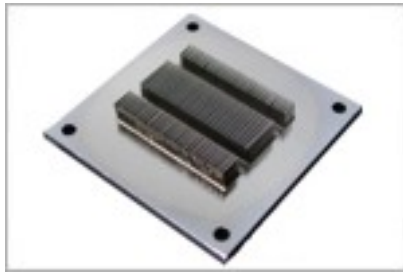
Figure 5. Schematic arrangement and flow conditions for a refrigerated air cooler to remove nominal 200 W of heat load

The conclusion from this simulation is that the effective coefficient of performance for this system is approximately 2. That is, a parasitic power of 93 W must be expended to remove the 200 W heat load, with an overall increase in the temperature-driving potential by a factor of 2 (from that for the previous case discussed). The improvement in the heat exchanger requirement is not substantial to offset the increase in parasitic power and the complexity for this system.

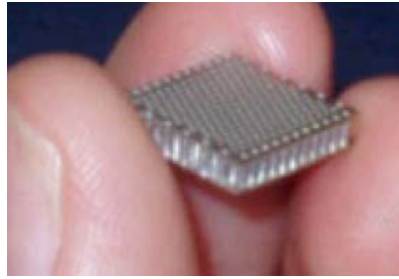
Microchannels and Microjet Arrays

We attempted to find hardware suitable for use with air cooling. We found two types of devices that are currently being offered as cooling devices for personal computer CPUs. These are microchannel devices

and microjet arrays. Figure 6 (a) shows a CPU cooler internals with microchannels and 6 (b) shows a micro jet array. (Courtesy Preytek, Co, Inc, and International Mezzo Technologies, Inc., respectively).



6(a)



6(b)

Figure 6. Photographs of microchannel and microjet devices

Microchannels have been studied and many results are available in the literature. We attempted to look into the micro jet arrays in detail for this task. The manufacturer's literature provided experimentally measured data for the microjet devices as shown in Figure 7.

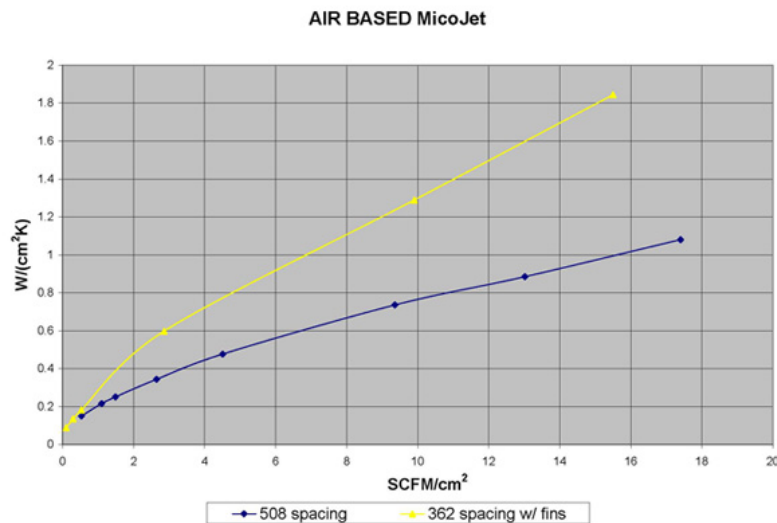


Figure 7. Variation of heat transfer coefficient with air flux (Courtesy: International Mezzo Technologies, Inc.)

An effective heat transfer coefficient is plotted as a function of air flux rate for these jets with differing jet-to-jet spacing. The abscissa can be converted to an effective superficial impingement velocity. The quantity 1 SCFM/cm² refers to a superficial impingement velocity of 4.72 m/s. Because the jets occupy only a fraction of the total area and the actual volumetric flow might be different from the standard volume flow quoted, the actual impingement velocity is likely to be a lot higher. The heat transfer coefficient from this device is a factor of 250 higher than typical values nominally encountered in industrial applications. If such high heat transfer coefficient can be verified, this device might have the

potential to fill the need for use with power electronics. Based on this assessment, we attempted to model the flow and heat transfer using this device.

CFD Analyses of Microjets

Simple axisymmetric airflow analysis of a microjet is shown in Figure 8. The jet diameter is fixed at 300 μm and the target diameter is 500 μm . Airflow rate is 5 CFM/ cm^2 . Figure 8 shows the airflow pattern at an air impingement velocity of 270 m/s.

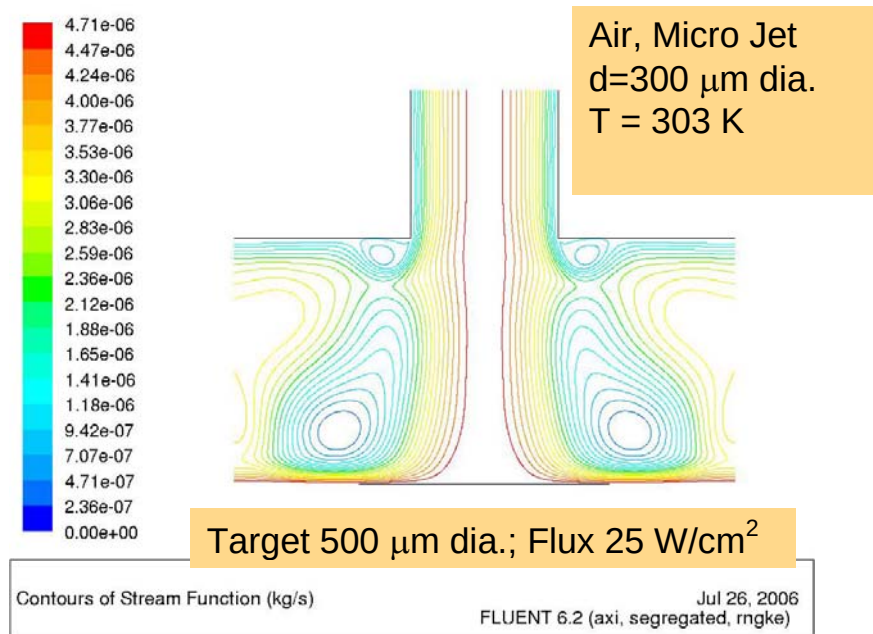


Figure 8. Streamline flow pattern for an impinging air jet

Figure 9 shows the temperature contours. Temperatures are indicated in Kelvin. Based on the heat flux of 25 W/ cm^2 , and an overall maximum temperature difference of 15 K between the target hot surface and the inlet air, we arrive at a nominal heat transfer coefficient of 1.33 W/ m^2K . With an average target temperature of 51.25°C and inlet air temperature of 26.85°C, the average heat transfer coefficient is 1.07 W/ cm^2K .

Based on this simulation, it appears that these types of microjets at high air velocities have the potential to yield high heat transfer coefficients. Our work in FY07 will investigate these devices both analytically and experimentally to assess the potential.

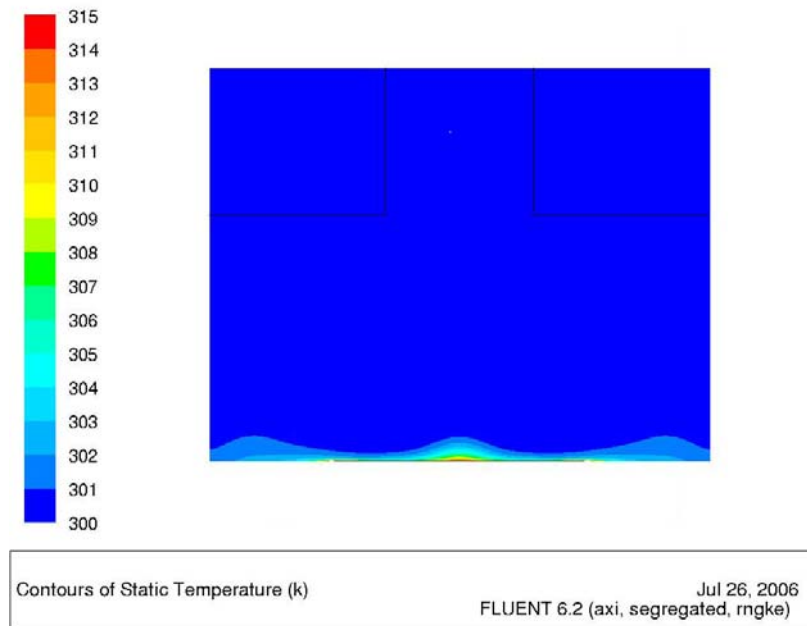


Figure 9. Temperature contours for the air jet impinging on a hot surface

Further analyses of the jet array as shown in Fig. 6b were carried out to assess their potential. Table 2 summarizes the results.

Table 2. Results of the CFD Analyses to Assess Heat Transfer Performance of Air Microjet Arrays

Configuration	Air Pressure Drop (kPa)	Wall Maximum Temperature (°C)	Effective Heat Transfer Coefficient (W/cm ² K)
1 CFM/cm ² 1.5 mm pitch	24	114	0.36
5 CFM/cm ² 0.6 mm pitch	11	124	0.27
5 CFM/cm ² 1.5 mm pitch	417	59	0.97

Based on these analyses we find that a heat transfer coefficient of 0.5 W/cm²K is readily achieved with these devices. We, however, need to verify this performance based on experiments scheduled in the future.

Scope and Limitations

Reported results in this paper are confined to data obtained from the literature and CFD analyses conducted at NREL. We will attempt to get relevant data for validation of these results when our laboratory will be in full operation during the next fiscal year.

Technical Discussion

Critical applications for air cooling occur in applications related to gas turbine blade cooling. Extremely high-heat fluxes must be removed to maintain a lower temperature at the turbine blades. There is a significant effort going on in the gas turbine industry, federal labs, and universities. Cooling for power electronic components can benefit from many of these studies related to blade cooling.

As an example of such application, Marques and Kelly [2004] report on fabrication and performance of micro pin fin arrays. Their studies use 500 μm diameter fins in air passages that is 500 μm apart. Air-side heat transfer coefficients of up to 0.8 W/cm²K have been reported. They also provide correlations for Nusselt numbers and friction factors that can be used to optimize the geometry for a particular application. Many other cooling strategies for blade cooling are found in Hans et. al [2001] and Shen et. al. [1996]. We believe a thorough literature review and an extended experimental study will result in viable options for cooling power electronic chips with air.

Conclusions

Air cooling is likely to meet the program goal for heat flux removal at a rate of 200 W/cm² with relatively low cost and complexity. While air cooling may not be appropriate for the current generation of silicon-based devices, it remains an attractive option for devices that can operate at higher temperatures. Use of air helps avoid the need for a secondary cooling fluid in vehicles. Improving chip manufacturing technologies continue to push the envelope on the highest temperature environment that these chips may see during operation.

Air cooling remains a viable option for power electronics. Continued research in this area will result substantive progress in this cooling technology. We recommend maintaining an effective R&D effort in this area.

References

- Hans, J.C.; Dutta, S.; Ekkad, S., Gas Turbine Heat Transfer and Cooling Technology, Taylor and Francis, 2001.
- Marques, C.; Kelly, K.W., "Fabrication and Performance of a Pin Fin Micro Heat Exchanger," J. Heat Transfer, v126, pp 434-444, June 2004.
- Shen, J.R.; Wang, Z.; Ireland, P.T.; Jones, T.V.; Byerley, A.R. "Heat Transfer Enhancement Within a Turbine Blade Cooling Passage Using Ribs and Combinations of Ribs With Film Cooling Holes," ASME J. of Turbomachinery, v118, pp 428-434, 1996.

2.4 Modeling Two-Phase Spray and Jet Impingement Cooling

Principal Investigator: Sreekant Narumanchi

National Renewable Energy Laboratory

MS 1633

1617 Cole Blvd

Golden, CO 80401-3393

Voice: 303-275-4062; Fax: 303-275-4415; E-mail: sreekant_narumanchi@nrel.gov

DOE Technology Development Manager: Susan A. Rogers

Voice: 202-586-8997; Fax: 202-586-1600; E-mail: Susan.Rogers@ee.doe.gov

Abstract

Meeting the FreedomCAR goals of cost, weight, and volume of power electronics components requires efficient thermal control strategies to dissipate the heat generated during the operation of these devices. Several parallel paths are being pursued to achieve the goal of 250 W/cm² heat dissipation while keeping the silicon die (in the IGBT package in an inverter) temperature below 125°C. This report describes both modeling and experiments, in single and two-phase jet/spray cooling in the context of cooling power electronics devices such as IGBTs in inverters of hybrid vehicles. Single-phase self-oscillating jets are being investigated experimentally. Significant progress has been made in modeling boiling jet impingement cooling of the IGBT package using R134a. An experimental apparatus has also been fabricated to enable two-phase jet/spray impingement experiments with fluids such as R134a, HFE7100/7200, and a glycol-water mixture.

Objectives

In hybrid vehicles, inverters are used for DC-AC conversion. These inverters involve a number of insulated gate bipolar transistors (IGBTs), which are used as on/off switches. The heat dissipated in these transistors can result in heat fluxes of up to 250 W/cm², which makes the thermal management problem quite important. The broad objective of this task is thermal control of automotive power electronics components. The FreedomCAR goal is to dissipate 200–250 W/cm² from the silicon die in the IGBT package. It is also desired for the maximum die temperature to be below 125°C. The automotive industry would like to eliminate any additional cooling loops. Hence, it is preferable to use a glycol-water mixture drawn from the engine cooling outlet at 105°C as the coolant for the power electronics components. Accordingly, the specific objectives of this task are to explore liquid jets and sprays, both in the single-phase and two-phase regime for cooling the IGBT package and meeting program goals. With single-phase jets, concepts such as self-oscillating jets are also explored. Jets and sprays in the boiling regime are also investigated.

Approach

NREL is exploring the performance of both single-phase as well as two-phase jets and sprays. The rationale is to characterize the heat transfer performance of these technologies in the context of cooling automotive power electronics and establish a database. Ultimately, this information will be fed into overall system-level modeling in order to ascertain the cooling technologies most suited for automotive applications.

To meet the objectives outlined above, the following approach was adopted in FY 2006:

- **Experiments with steady and self-oscillating single-phase jets**

One of the FreedomCAR goals is to utilize glycol water at 105°C for cooling the automotive power electronics components. Given that it is desired to maintain the silicon die at 125°C, it implies only a 20°C difference between the die and the coolant temperature. This implies that high heat transfer coefficients would be required in conjunction with concepts like enhanced surfaces. There have been some studies in the literature which demonstrate that oscillating jets could enhance heat transfer over steady jets by as much as 50%–70% [1-4]. As a baseline, for a 1.25 mm diameter steady freesurface water jet at a velocity of 10 m/s, a heat transfer coefficient of 40,000 W/m²K can be obtained. In some of these studies [1-3], the oscillation is induced by mechanical means, while in others [4] the jets are self-oscillating. Self-oscillating jets involve no moving parts and the oscillations are induced strictly by pressure differences in the fluid. This makes them quite attractive. However, there is very limited literature on the heat transfer performance of self-oscillating jets. Hence, we decided to explore the heat transfer potential of self-oscillating jets.

Bowles Fluidics Corporation (BFC) has a long history of fabricating fluidic nozzles for a variety of applications. NREL established a CRADA with BFC to explore the heat transfer performance of different types of fluidic nozzles. We fabricated an apparatus which allows characterization of the heat transfer performance of different types of fluidic nozzles. Different fluidic nozzles have been obtained from BFC and some initial heat transfer experiments have been performed to characterize the performance of these nozzles. Comparable steady jet experiments have also been performed to serve as a baseline against which to compare the performance of the fluidic nozzles. Initially, water is being used as the coolant; however, the ultimate goal is to use a glycol-water mixture at 105°C as the coolant. Results of the heat transfer performance will be fed into our thermal systems model to determine the impact of these technologies at the system level.

- **CFD model of nucleate boiling in impinging jets**

Using glycol-water jets at 105°C inlet temperature implies that in order to keep the silicon die temperature at 125°C, very high heat transfer coefficients may be required (almost 140,000 W/m²K without any surface area enhancements) in order to dissipate 250 W/cm² from the IGBT package. At present, it is not very clear that such heat transfer coefficients can be achieved with the use of glycol-water jets without resorting to very high jet velocities (in excess of 35 m/s). Hence, in parallel, we are also exploring two-phase jet impingement cooling of the IGBT package. In electronics cooling, for jets in the two-phase regime, it is desirable to operate in the nucleate boiling regime [5] since a small increase in the target surface temperature is accompanied by a large increase in the heat flux that can be dissipated from the surface.

Therefore, in FY 2005 NREL employed the Eulerian multiphase model, within the framework of the CFD code FLUENT, to study nucleate boiling in impinging jets. A user-defined function was incorporated into FLUENT to enable these predictions. A mechanistic model of nucleate boiling was used. However, the vapor energy equation was not solved and it was assumed that the vapor is at the saturation temperature. In FY 2006, the model was improved to solve for the energy equation of the vapor. The model was validated with a couple of experimental studies reported in the literature on nucleate boiling in submerged jets. The code was then employed to study the cooling of an IGBT package using impinging R134a jets.

R134a and HFE7100/7200 are fluids that are suitable for use in electronic cooling applications. They have the potential to meet the targets of 250 W/cm^2 heat dissipation from the silicon die while keeping the die temperature below 125°C using jet velocities of the order of 10 m/s .

- **Apparatus for spray and boiling jet experiments**

Concurrent with the numerical simulations of boiling jets, a subcontract was established with Mudawar Thermal Systems in FY 2006 to construct an apparatus which will enable spray and jet experiments in the two-phase regime. Two test vessels are being constructed – one of which will be able to withstand pressures of up to 600 psi. The intent is to perform boiling jet and spray experiments with R134a and HFE7100/7200. As mentioned previously, both R134a and HFE7100/7200 are fluids which are suitable for automotive applications. The experiments with R134a are to be performed at pressures of up to 250 psi, while the experiments with HFE7100/7200 will be performed at atmospheric pressure ($\sim 15 \text{ psi}$). This is the reason for the fabrication of two separate test vessels.

The vessel that can withstand high pressures requires thicker walls; however, this also means that the experiments would require a longer time to reach steady state. Hence, for low pressure experiments, the test vessel with thinner walls will be used. The fabrication of the test vessels, the target surface, heaters and the rest of the instrumentation is reaching completion. All experiments will be performed on a target surface of area $1 \text{ cm} \times 1 \text{ cm}$. The intent is to first characterize the heat transfer performance of R134a and HFE7100/7200 sprays and jets on a test surface in a simple configuration and, if promising heat transfer performance is observed, test these cooling schemes on an actual IGBT package.

Major Accomplishments

The main accomplishments of FY 2006 are listed below.

- In FY 2005, we performed extensive CFD modeling of single-phase jets (both freesurface and submerged) and characterized their heat transfer performance. The information acquired from these modeling efforts was utilized to design a jet impingement cooling scheme for a low resistance IGBT package. This will be described in much more detail in another milestone report.
- The high heat flux (HHF) test loop in the Electrical Systems Laboratory at NREL was completed. This loop has all the components necessary to enable single-phase liquid cooling experiments. Broadly, the loop consists of a reservoir, pump, filter, flowmeters, preheater (to heat the working fluid), temperature control units, and several outlets for plugging in thermocouple sensors. All measurements (e.g., pressure, liquid flowrate, temperatures) are made through a data acquisition system via Labview. The laboratory also has a power supply unit (ABC1000) that can supply power for a variety of applications.
- The heat transfer performance of self-oscillating water jets was experimentally explored using fluidic nozzles provided by BFC. Initial results indicated that at a certain flow rate and nozzle-to-target distance, the heat transfer coefficients due to these self-oscillating freesurface jets is almost 30% higher than steady freesurface jets, given the same nozzle exit area and jet velocity. More detailed tests employing concepts from design of experiments are currently underway to evaluate different

fluidic nozzles. The goal is to determine the optimal jet configuration and the subsequent performance.

- In active collaboration with Fluent Inc., a user-defined function was implemented in FLUENT, which enables numerical simulations of impinging jets involving nucleate boiling. The code was applied to study jet impingement cooling, via nucleate boiling, of an IGBT package using R134a jets. Conditions were demonstrated for dissipating 200 W/cm^2 while keeping the silicon die temperature close to 125°C . These are important results because they demonstrate conditions under which R134a can be used to come close to meeting the FreedomCAR goals. Some initial results have been published at the ITherm conference [6] and detailed results will be submitted for possible publication in a journal [7].
- Mudawar Thermal Systems, under subcontract with NREL, has completed fabrication of a test vessel and apparatus that can be used to perform jet and spray cooling experiments in the two-phase regime. These will enable experiments with R134a and HFE7100/7200, two fluids that are particularly suitable for use in automotive applications.

Technical Discussion

The section outlines the progress made in FY 2006. It elaborates on the major accomplishments outlined in the previous section. This section is divided into three parts – one for single-phase jet experiments, one for two-phase jet modeling, and the third for experiments with two-phase sprays/jets.

• Single-phase jets

As mentioned previously, one of the FreedomCAR goals is to use a glycol-water mixture as a coolant at 105°C in the single-phase regime. Hence, we have established a test loop at NREL that will enable experiments with single-phase jets. This loop is shown in Figure 1(a). It consists of a reservoir, pump, filter, mass flowmeters, turbine flowmeters, and valves for regulating flow rate. The loop is also fully instrumented with pressure sensors, thermocouples for monitoring temperatures, and a data acquisition system controlled by LabView.

We also fabricated an apparatus that would enable characterization of the heat transfer performance via single-phase jets. The apparatus is shown in Figure 1(b). It consists of two Teflon plates with a transparent polycarbonate cylinder in between. A copper plug 1-1/2 inches in length and half an inch in diameter fits snugly in the center of the lower Teflon plate. The surface of the copper plate serves as the target surface on which the jet impinges. We are exploring self-oscillating jets to enhance heat transfer coefficients from the target surface. Figure 1(c) shows the sample nozzles obtained from Bowles Fluidics. Six different types of fluidic nozzles are shown. Initial testing has been carried out with two different types of fluidic nozzles and one steady jet. The area of the jet exit of the fluidic nozzles is $1.1 \times 1.1 \text{ mm}^2$. The steady jet producing nozzle has the same exit area of 1.21 mm^2 and has a circular opening of 1.24 mm diameter. For the steady jet, a circular opening was chosen to make the steady jet nozzle very efficient. The rationale was that this would provide a fair basis for comparison with the performance of the fluidic nozzles. Figure 1(d) shows a snapshot of one of the experiments

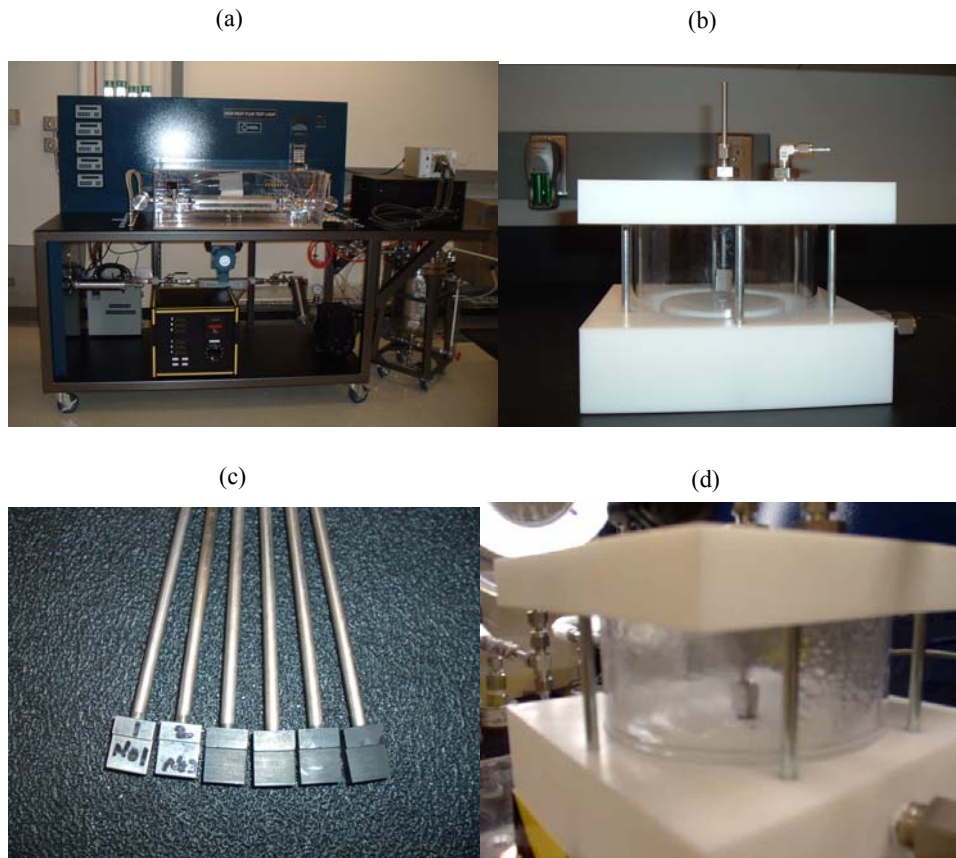


Figure 1. (a) Test loop; (b) experimental apparatus; (c) nozzles; (d) one sample experiment

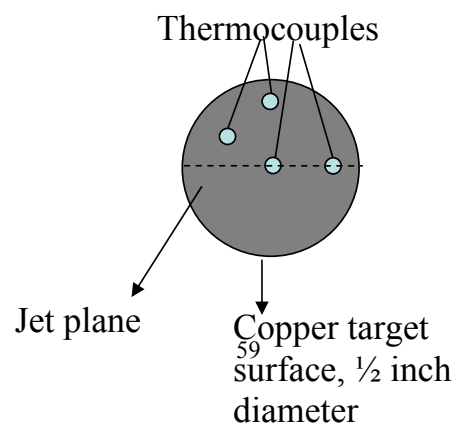


Figure 2. Surface of the copper plug that serves as the target surface

The surface of the copper plug is shown in Figure 2. The jet centerline is aligned with the center point of the target. For self-oscillating jets, the plane of oscillation is also shown in the figure. There are four thermocouples on the surface of the copper plug. The thermocouples are brought out through the backside of the apparatus. The rationale for placing multiple thermocouples is to determine the temperature distribution on the target surface. At the back of the copper plug, a resistive heater from RF Florida Labs is attached via a couple of bolts. Thermal grease is placed between the copper plug and the resistive heater to ensure good contact. A thermocouple is also attached to the heater in order to monitor its temperature. The power is supplied to the heater via ABC 1000 power supply (from Aerovironment Inc.) The maximum operating temperature of the heater is rated at 150°C. Key measurements being made include the jet inlet temperature, the jet outlet temperature, temperatures at four points on the copper target surface, the temperature of the resistive heater, the jet flow rate, the pressure drop across the jet, and the power input to the heater.

A heat transfer analysis was done (in FLUENT) on the lower Teflon plate to gauge the heat loss from the sides and the back of the apparatus when heat is applied to the resistor. A heat transfer coefficient was applied to the top of the copper target surface and the surrounding Teflon block. A wide range of heat transfer coefficients characteristic of impinging jets were applied on the copper surface (about 20,000–50,000 W/m²K). From the simulations, it is expected that the heat losses from the sides and back of the apparatus should be no more than 3%. The apparatus is placed on an insulating material with a very low conductivity of 0.02 W/mK. The apparatus allows for both freesurface and submerged configurations.

In the literature, the three main categories of jets that have been explored are freesurface, submerged, and confined submerged jets. These are illustrated schematically in Figure 3 (taken from [8]). In a freesurface configuration (Figure 3(a)), the liquid jet issues into a different fluid—usually a gas which is of much lower density than the liquid, while in a submerged jet the liquid jet issues into another fluid of comparable density (Figure 3(b)). In a submerged confined jet (Figure 3(c)), the liquid jet is in a submerged configuration, and in addition there is also a plate/wall confining the liquid after impingement on the target surface.

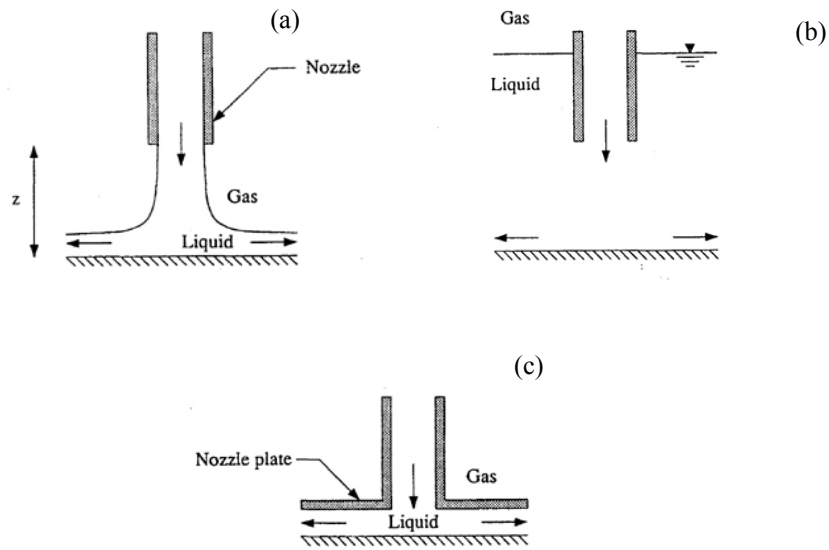


Figure 3. Different jet impingement configurations (taken from [8]): (a) free-surface jet, (b) submerged jet, (c) confined submerged jet

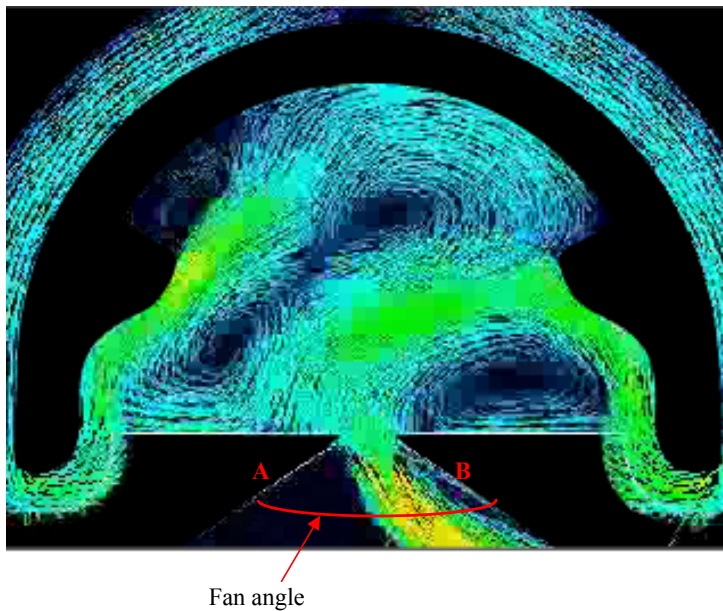


Figure 4. Illustration of the principle behind self-oscillating jets (photo courtesy of BFC)

As mentioned earlier, NREL is exploring the performance of self-oscillating jets. Figure 4 illustrates the concept behind self-oscillating jets. The jet oscillates within the angle depicted in Figure 4. This angle is referred to as the fan angle. The frequency of these oscillations is determined by the time taken for the jet to detach from one wall (wall A in Figure 4), attach to wall B, detach from wall B, and come back and attach to wall A. Essentially, the jet performs a sweeping motion within the fan angle. From a heat transfer viewpoint, this sweeping motion has the potential to disrupt the boundary layer growth and thereby enhance heat transfer between the surface and the liquid.

The initial tests were carried out with two different types of fluidic nozzles: one with a Strouhal number (defined as $St = fd/v$, where f is the frequency of oscillation, d is the hydraulic diameter of the jet and v is the velocity of the jet) of 0.1 and the other with St of 0.02. As mentioned before, both these jets have an opening of dimensions 1.1 x 1.1 mm. The baseline steady jet has a diameter of 1.24 mm and the same area of 1.21 mm² as the fluidic nozzles. The frequency vs. jet velocity and the fan angle vs. velocity characteristics for fluidic1 are shown in Figure 5, while the frequency vs. velocity characteristic for fluidic2 is shown in Figure 6. The frequency vs. velocity characteristic is almost a linear curve for both fluidic nozzles within the velocity range of interest of 3–10 m/s.

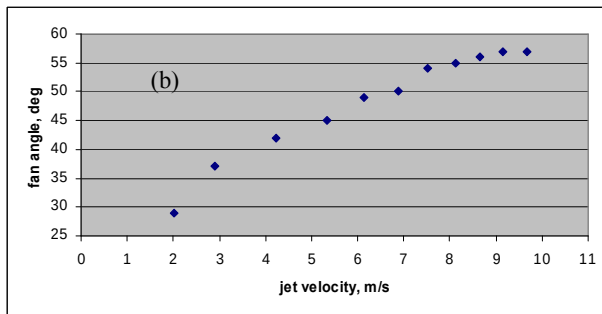
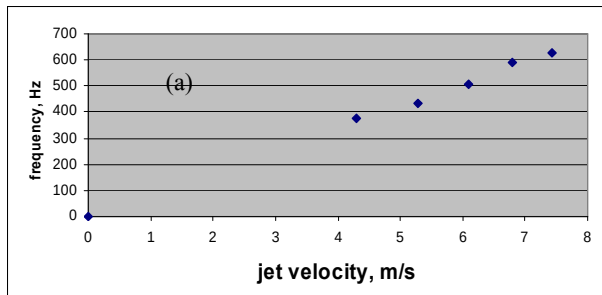


Figure 5. (a) Frequency versus velocity characteristics for nozzle 1 (called fluidic1) with $St \sim 0.1$; (b) fan angle versus velocity characteristic

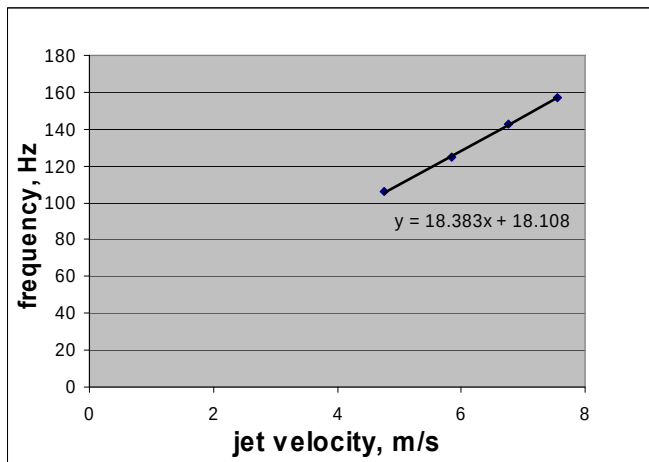


Figure 6. Frequency versus jet velocity characteristic for nozzle 2 (fluidic2) with $St \sim 0.02$

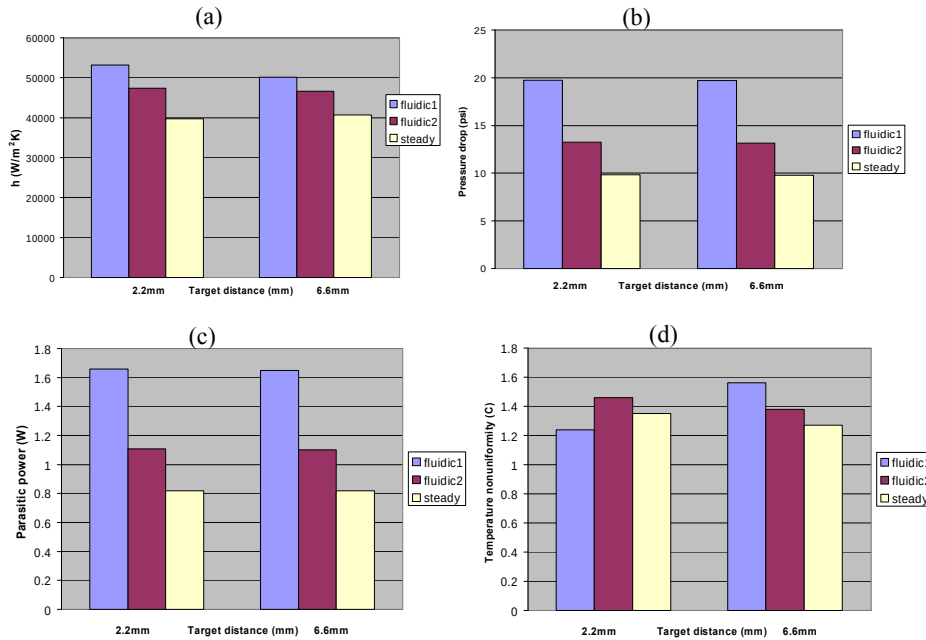


Figure 7. Results for the different nozzles at two target distances of 2.2 mm and 6.6 mm at a velocity of 10 m/s (0.73 lpm). (a) heat transfer coefficients; (b) pressure drop; (c) parasitic power; (d) temperature nonuniformity

Table 1. Summary of the Freesurface Jet Experimental Results With the Three Different Nozzles, Including the Steady Jet Nozzle

Nozzle	Sn _p (mm)	Q (lpm)	ΔP (psi)	Power, P (W)	h (W/m ² K)	$\Delta T_{\text{nonunif}}$ (C)	h/P (1/m ² K)
fluidic1	2.2	0.733	19.76	1.66	53,162	1.24	32,025
fluidic1	6.6	0.732	19.7	1.65	50,133	1.56	30,384
fluidic2	2.2	0.735	13.24	1.11	47,362	1.46	42,669
fluidic2	6.6	0.731	13.17	1.1	46,672	1.38	42,429
steady	2.2	0.73	9.83	0.82	39,681	1.35	48,391
steady	6.6	0.73	9.79	0.82	40,674	1.27	49,602

Table 1 shows a summary of the preliminary freesurface jet experiments performed with fluidic nozzles and the steady jet. Two different nozzle-exit-to-target distances were considered—2.2 mm and 6.6 mm at a velocity of 10 m/s (corresponding to 0.73 lpm). In the table, Sn_p denotes the nozzle exit-to-target distance, Q is the jet flow rate in lpm, ΔP is the pressure drop across the jet (psi), P is the parasitic power consumption (product of Q and ΔP), h is the heat transfer coefficient, $\Delta T_{\text{nonunif}}$ is the nonuniformity in the temperature on the copper surface (i.e., the maximum temperature difference among the four thermocouple readings), and h/P is a measure of the performance of the jet. The last column in Table 1 indicates the type of nozzle being used. Figure 7 is basically a visual representation of the results presented in Table 1. It shows the heat transfer coefficients (Figure 7(a)), the pressure drop across the jet

(Figure 7(b)), the parasitic power consumption (Figure 7(c)), and the temperature nonuniformity (Figure 7(d)) on the target surface. The heat transfer coefficient is defined as follows:

$$h = \frac{q}{A(T - T_{\text{ref}})} \quad (1)$$

where h is the heat transfer coefficient, q is the power applied to the resistive heater, A is the area of the surface of the copper plug (1.267 cm^2), T is the surface temperature of the copper surface (average of the four thermocouple readings on the copper target surface), and T_{ref} is the jet inlet temperature. For the initial experiments reported here, T_{ref} is at 25°C .

Figure 7(a) shows the heat transfer coefficients at the two target distances of 2.2 and 6.6 mm, and a jet velocity of 10 m/s (flowrate of 0.73 lpm). Figure 7(a) shows that fluidic1 performs better than the steady jet for both target distances. At the smaller target distance of 2.2 mm, the enhancement in heat transfer due to fluidic1 is about 30%. At a velocity of 10 m/s, the frequency of fluidic1 is about 910 Hz, while the frequency of fluidic2 is 272 Hz. It is postulated that at the close target distances, the high frequency of fluidic1 combined with the elevated velocity is sufficient to disrupt the boundary layer growth on the target surface.

The time-averaged boundary layer for this case is different from that of the time-averaged boundary layer for the steady jet at similar velocities. This could possibly explain the enhancements observed. The heat transfer performance of fluidic2 lies between fluidic1 and the steady jet. Figure 7(b) shows that the pressure drop associated with fluidic1 is quite high—almost twice that of the steady jet. Accordingly, Figure 7(c) shows that the parasitic power consumption (product of the flow rate and the pressure drop across the jet) of fluidic1 is highest (twice that of the steady jet). Figure 7(d) shows the temperature nonuniformity on the copper surface. There is no specific trend, and all the non-uniformities range between 1.2°C and 1.5°C . The center point (stagnation point) temperature is the lowest—in line with trends reported in the literature [9-11], and the remaining three thermocouples are bunched together within 0.5°C of one another.

The goal is to increase the heat transfer coefficient at a given flow rate, but the goal is also to reduce the parasitic power. Hence, the goal is to increase the metric h/P , which is tabulated in Table 1. Interestingly, the numbers for h/P reveal that the steady jet still has the highest h/P , which suggests that the pressure drop for fluidic1 is very high. Fluidic2 again lies between fluidic1 and the steady jet. At a velocity of 10 m/s, we have been able to achieve heat transfer coefficients of the order of $55,000 \text{ W/m}^2\text{K}$ with freesurface self-oscillating jets. Currently, we are in the process of testing fluidic nozzles, which have much lower pressure drop while maintaining the frequency characteristics of fluidic1.

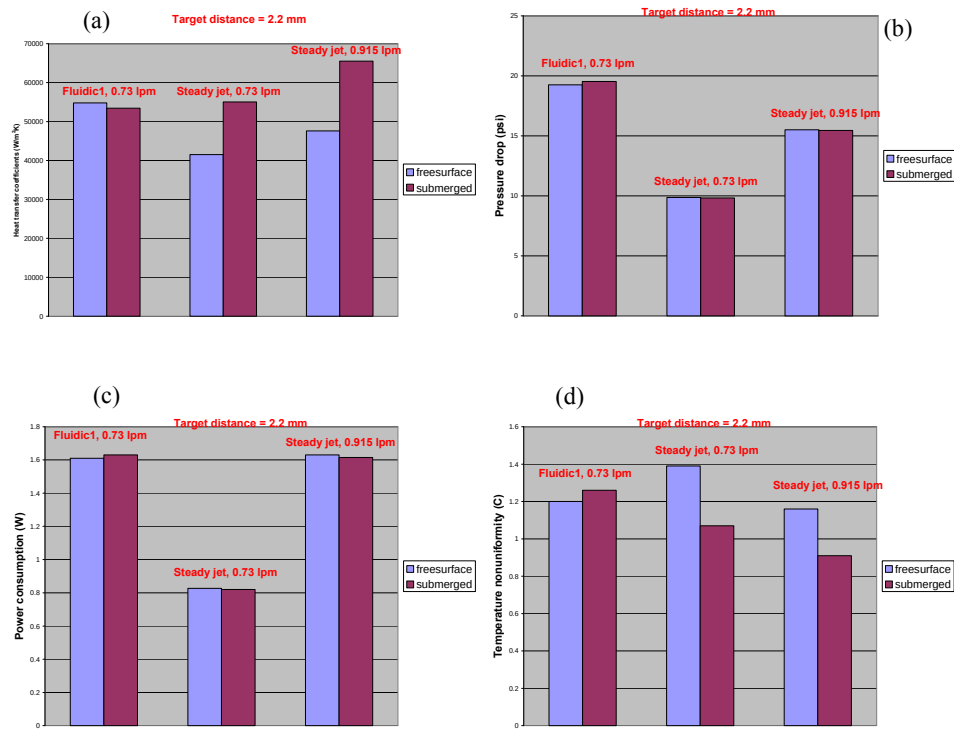


Figure 8. Comparison between freesurface and submerged jets for fluidic1 and the steady jet nozzle, (a) heat transfer coefficient; (b) pressure drop; (c) power consumption; (d) temperature nonuniformity

Table 2. Comparison Between Freesurface and Submerged Jets for Fluidic1 and the Steady Jet at 2.2 mm Target Distance and 10 m/s Velocity (Flowrate ~ 0.73 lpm)

SnP (mm)	Q (lpm)	ΔP (psi)	power (W)	h (W/m²K)	$\Delta T_{nonunif}$ (C)	h/P (1/m²K)	Nozzle
2.2	0.73	19.25	1.61	54,797	1.2	34,035	fluidic1-freesurface
2.2	0.732	9.87	0.827	41,513	1.39	50,197	steady-freesurface
2.2	0.919	15.5	1.63	47,583	1.16	29,192	steady-freesurface
2.2	0.73	19.53	1.63	53,448	1.26	32,790	fluidic1-submerged
2.2	0.73	9.83	0.82	55,022	1.07	67,100	steady-submerged
2.2	0.913	15.45	1.615	65,529	0.91	40,575	steady-submerged

In addition to freesurface jets, some submerged jet experiments have also been performed. The experiments performed and the results are summarized in Table 2. The experiments were performed with the steady jet and fluidic1 at a target distance of 2.2 mm and a velocity of 10 m/s (flowrate of 0.73 lpm). The various quantities listed in Table 2 are similar to those in Table 1. Figure 8 shows the heat transfer coefficients (Figure 8(a)), pressure drop (Figure 8(b)), parasitic power consumption (Figure 8(c)), and temperature nonuniformity (Figure 8(d)). With the steady jet, in an attempt to make the parasitic power consumption identical to that of fluidic1, the flow rate was increased to 0.915 lpm and experiments were performed for both freesurface and submerged configurations—everything else remaining the same. The freesurface results are the same as shown in Figure 7 and Table 1.

Interestingly, for the freesurface configuration, when the flow rate for the steady jet is increased to 0.915 lpm in order to make the power consumption the same as that of fluidic1, the steady jet still gives about 14% lower heat transfer coefficient than fluidic1. However, the situation for submerged jets is quite different. The submerged steady jet gives comparable heat transfer coefficients to the submerged fluidic1 jet at the same flow rate of 0.73 lpm. When the flow rate of the steady submerged jet was increased to 0.915 lpm in order to make the power consumption the same as the submerged fluidic1 jet, the heat transfer coefficient of the submerged steady jet is about 20% higher than the fluidic1 submerged jet. The key result is that the heat transfer coefficient due to the steady submerged jet is about 30% higher than the steady freesurface jet.

This result is somewhat in line with trends reported in the literature. A few studies have found that the submerged steady jet yields higher heat transfer coefficients than the freesurface steady jet at elevated flowrates/velocities [12]. One postulated reason is the splashing associated with freesurface jets and the thinning of the fluid on the target surface and the consequent bulk warming of the fluid due to the boundary layer reaching the surface of the fluid [12]. Also, it is worth noting that at very close target distances (2.2 mm), given the shape of the nozzle housing (Figure 1(c)), the submerged steady jet configuration becomes more like a submerged confined jet configuration (Figure 3(c), [13]), which has been found to give higher heat transfer performance than a regular submerged steady or a freesurface steady jet [13].

Table 2 also shows h/P values for the different cases. For the freesurface configuration, when the parasitic power consumption is the same, it can be seen that fluidic1 has a higher h/P value (34,035) than the steady jet (29,192). However, for the submerged configuration, when the parasitic power consumption is the same, fluidic1 has a lower h/P value (32,790) than the steady jet (40,575).

The results reported here are initial results. Additional fluidic nozzles have been acquired from Bowles Fluidics. A more detailed set of experiments is being planned utilizing concepts from design of experiments. A detailed uncertainty analysis will also be performed in order to assign error bands to the heat transfer results as well as all other relevant experimental quantities. Also, these experimental results have been reported with water. If significant benefits are found from the use of self-oscillating jets, experiments will also be performed with glycol-water at a temperature of 105°C.

Impact of fluid heat transfer coefficient and fluid temperature

In this section, we will examine the impact of fluid heat transfer coefficient as well as fluid temperature via an ANSYS 3D conduction model with appropriate boundary conditions. The domain is depicted in Figure 9. Figure 9(a) shows the full domain, which is basically the IGBT half bridge, while Figure 9(b) shows a view of part of the half bridge. Figure 9(b) shows the different layers involved. Figure 9(c) shows the backside of the structure. A portion of the aluminum plate is cut out and heat transfer coefficients (h) are imposed directly on the exposed copper layer. For each cutout, on the surrounding aluminum faces,

half the heat transfer coefficient is imposed ($h/2$). The choice of $h/2$ on the aluminum walls is appropriate as shown by detailed CFD modeling [14]. Curves for maximum die temperature vs. the imposed heat flux on the silicon die are obtained with different heat transfer coefficient values (Figure 10).

Figure 10(a) shows that to meet program goals (125°C die temperature with 250 W/cm^2 heat dissipation in the die) with 105°C inlet coolant temperature, very high heat transfer coefficient values will be required. Even a heat transfer coefficient of $140,000\text{ W/m}^2\text{K}$ brings the die temperature to only 140°C —much above the target temperature of 125°C . However, when the coolant inlet temperature is reduced to 70°C (Figure 10(b)), the program goals can be met with a heat transfer coefficient of $60,000\text{ W/m}^2\text{K}$. Figure 10 also gives a sense of what a heat transfer coefficient of $55,000\text{ W/m}^2\text{K}$ (a number which we got experimentally in an earlier section) implies in the context of the IGBT package cooling. In addition to attempts to increase the heat transfer coefficients, the surface area from which heat transfer takes place can also be enhanced. The challenge of using 105°C glycol-water as a coolant is a formidable one.

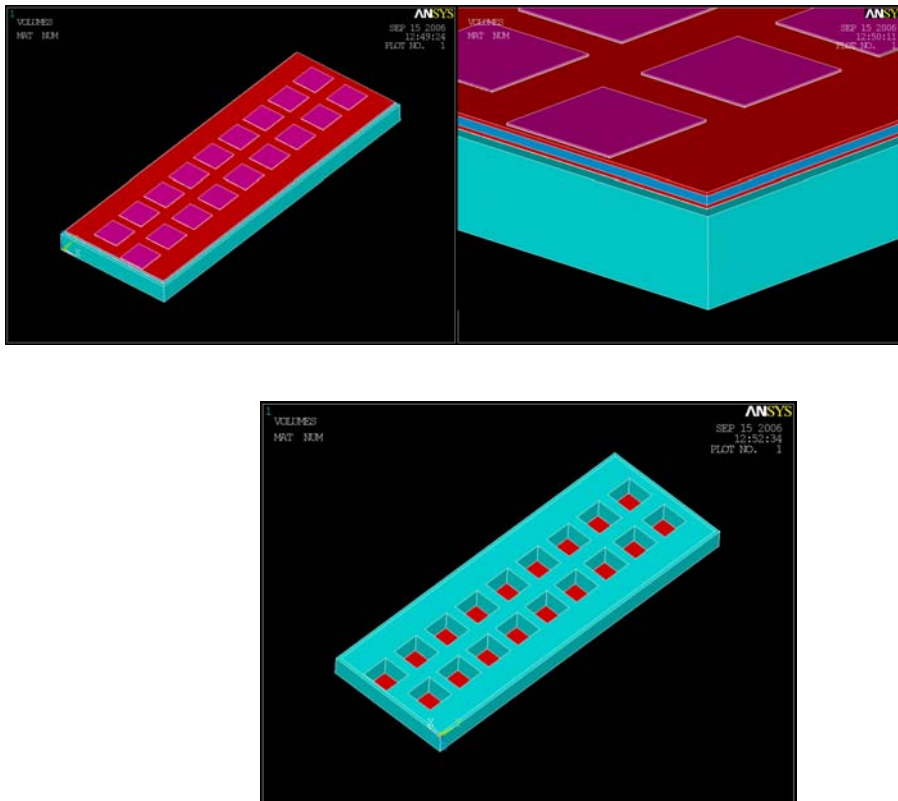


Figure 9. (a) IGBT half-bridge; (b) closeup showing the different layers; (c) bottom view of the IGBT half-bridge showing the cut through the aluminum plate

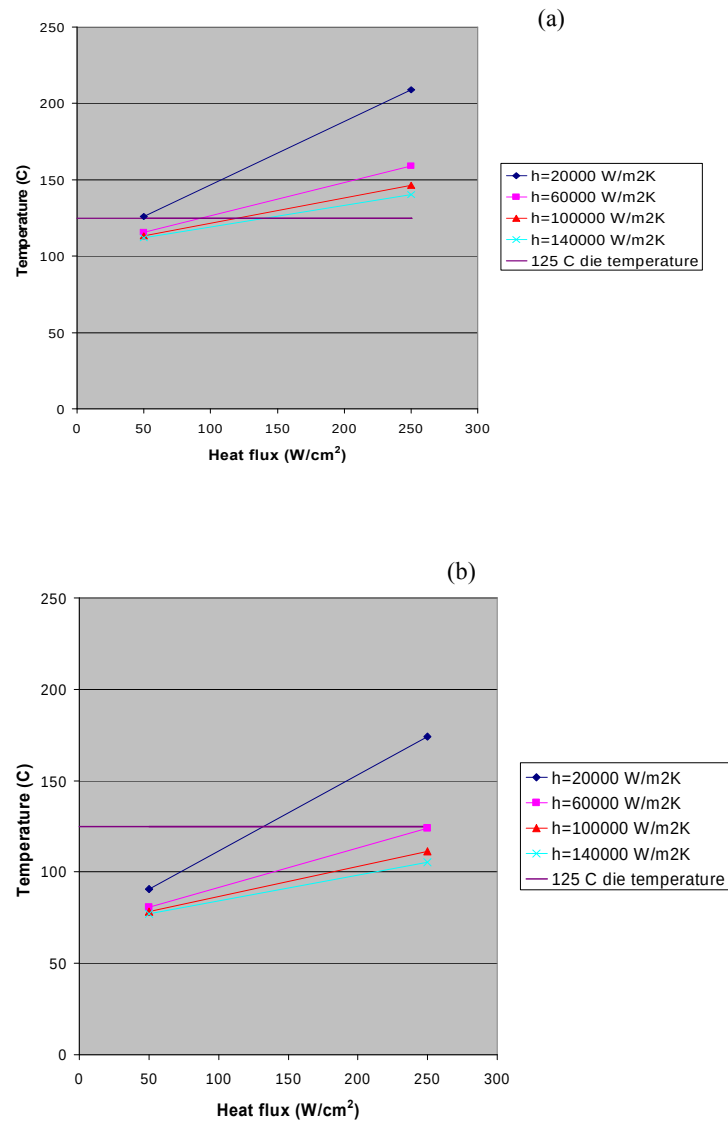


Figure 10. Die temperature versus heat flux for (a) $T_{\text{inlet}}=105^{\circ}\text{C}$; (b) $T_{\text{inlet}}=70^{\circ}\text{C}$

- Two-phase jets—numerical (CFD) modeling

As mentioned previously, it is not clear at this point if program goals can be met with 105°C glycol-water coolant. Hence, in parallel with the effort on single-phase jets, in FY 2006 work was also done on boiling jets (nucleate boiling). The rationale is to explore different cooling methodologies that could potentially meet the FreedomCAR goals and create a database of experimental and modeling results. These results can then be incorporated into system level modeling and design in order to ascertain the most feasible cooling solution in the automotive context.

In this section, turbulent jet impingement involving nucleate boiling is explored numerically. The framework for these computations is the CFD code FLUENT. For nucleate boiling, the Eulerian multiphase model is used. A mechanistic model of nucleate boiling is implemented in a user-defined function (UDF) in FLUENT. This effort has involved close collaboration with the staff at Fluent Inc. In the modeling effort in FY 2005 [6], the vapor phase was assumed to be at the saturation temperature, and the vapor energy equation was not solved. In FY 2006, the model was extended to solve the energy equation of the vapor. The model is described in detail in [7]. The numerical results for boiling water and R113 jets (submerged) are validated against existing experimental data in the literature.

R134a is already in use in automotive air-conditioning units. It is also a fluid which is suitable for automotive applications from various viewpoints. Hence, some representative IGBT package simulations that use R134a jets for cooling the package are also presented.

Experimental validations

In the following sections, we present two validations of our numerical modeling approach with experimental data available in the literature. These cases are chosen so as to span different fluid properties (water, R113), saturation temperatures, degree of subcooling, and jet velocities.

Validation With Experimental Study of Katto and Kunihiro [15]. First, we examine the experimental study of Katto and Kunihiro [15]. The domain and the boundary conditions are shown in Figure 11. A water jet with 3°C sub-cooling at atmospheric pressure (i.e., with $T_{\text{inlet}} = 97^\circ\text{C}$) impinges on a 10-mm-diameter disk with an inlet velocity of 2 m/s. The baseline nozzle diameter is 1.6 mm, and the distance between the nozzle exit and the heated plate is maintained at 3 mm. A heat flux is imposed on the hotplate surface, as shown in Figure 11. This is a submerged jet configuration. An axisymmetric domain is established. The RNG k-epsilon model with standard wall functions is used. With the use of the standard wall function, the y^+ close to the walls should be maintained above 30. All the results presented here are mesh-independent to within 5%. The properties of water at 1 atmosphere pressure are listed in Table 3.

There is ambiguity about the exact temperature measured in the experiments. All indications are that the stagnation point temperature is reported in the experiments. Figure 12 shows the boiling curve, which is a plot of the wall heat flux vs. the stagnation point wall superheat. The wall superheat is defined as $\Delta T_{\text{sat}} = T_w - T_{\text{sat}}$, where T_w is the wall stagnation point temperature while T_{sat} is the saturation temperature of the fluid at a given pressure. The experimental data are close to the stagnation point superheat (within 20%) from the CFD simulations, which is encouraging. Given the nature of this problem, uncertainties on the order of even 20% are quite common and acceptable. There is temporal fluctuation in all quantities—such as temperature, fluid volume

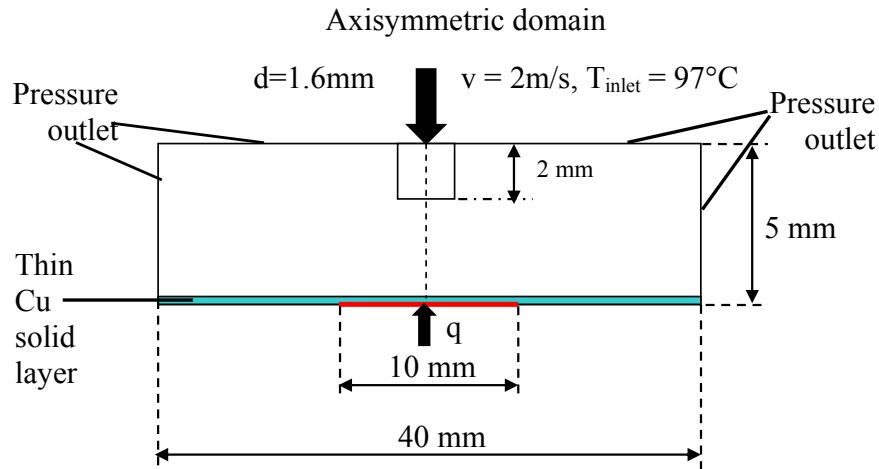


Figure 11. Axisymmetric domain used for the Katto and Kunihiro study (1973) with submerged water jets involving nucleate boiling.

Table 3. Properties of Water and R-113 at 1 Atmosphere
Pressure (1.013e+05 Pa)

	Water		R-113	
	Liq.	Vap.	Liq.	Vap.
Saturation temperature ($^\circ\text{C}$)	100		47.6	
Surface tension (N/m)	0.059		0.014	
Latent heat (J/kg)	2,257,000		144,000	
Density, kg/m^3	958	0.6	1,507	7.5
Specific heat, J/kg-K	4,219	2,010	980	724
Dynamic viscosity, Ns/m^2	2.83e-04	1.23e-05	5.23e-04	1.08e-05
Thermal conductivity, W/mK	0.68	0.025	0.074	0.01

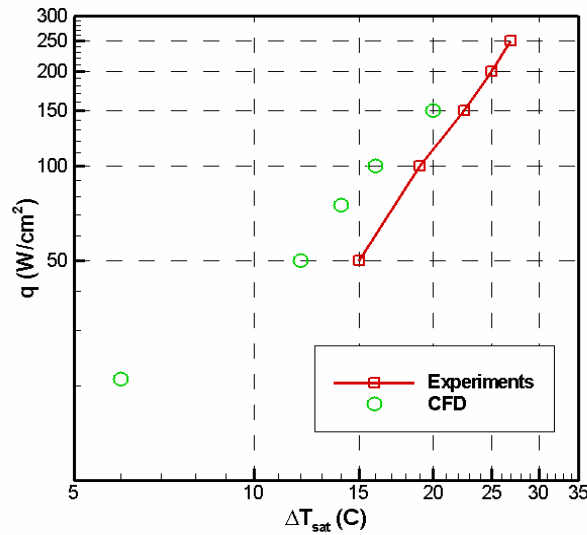


Figure 12. Boiling curve for the Katto and Kunihiro (1973) study

fraction, mass flow rates, and energy transfer rates obtained from the CFD simulations. A temporal average is reported here.

Validation with Zhou and Ma [16] Experimental Study. In this section, we consider another experimental study with submerged boiling jets. This is the study by Zhou and Ma [16] with a submerged R-113 jet. The rationale for choosing this study is that it involves a coolant distinct from water; it involves fluid subcooling and a wide range of jet velocities. The domain is shown in Figure 13. As a simplification, the domain is assumed to be axisymmetric. The nozzle diameter is 1 mm. The target plate (constantan foil) area in the actual experiment is 5 x 5 mm, corresponding to a disk of radius 2.8 mm. A heat flux is applied to the target surface. The distance between the end of the nozzle and the target surface is 5 mm. In the experiments, the stagnation zone temperature is measured with a spatial resolution of about 0.2 mm. Also, the direction of gravity is not the same as in the actual experiment. However, we have confirmed through simulations that the impact of gravity on nucleate boiling in impinging jets is not important. This aspect has also been demonstrated experimentally. Figure 14 shows the results of the validation.

The boiling curves (heat flux vs. stagnation point wall superheat) from both experiments and the CFD predictions are plotted in Figure 14. Figure 14(a) shows the curve for a jet velocity of 0.41 m/s while Figure 14(b) shows results for a jet velocity of 11.36 m/s. The R-113 properties at atmospheric pressure are listed in Table 3. The fluid subcooling is 18.5°C, which means that the jet inlet temperature is 302.3 K. At the elevated velocity, the saturation temperature of the fluid changes along the target wall; this aspect is accounted for in the code. The match between experiments and CFD predictions is good (they are within 10%). At the lower velocity, the wall superheat fluctuates quite a lot; hence, time-averaged quantities are reported. The results for wall superheat presented here are mesh-independent to within 2%. For both the velocities, the fraction of evaporative heat flux is plotted as a function of the applied heat flux. The fraction is simply the heat flux that goes into vaporizing the fluid divided by the total heat flux applied at the wall. As seen in Figure 14(a), at the lower velocity the fraction of evaporative heat flux is

quite high. It touches 75% at a flux of 54 W/cm^2 . Basically, as the jet velocity is decreased, we move closer to a pool boiling situation. On the other hand, for the higher velocity case (11.36 m/s), the fraction of evaporative heat flux is only 52% at a heat flux of 250 W/cm^2 .

For this higher velocity case, the prediction from a pure single-phase case is also shown, i.e., there is no boiling involved. At lower heat fluxes, it can be seen that there is virtually no difference between the single-phase and boiling predictions. However, as the heat flux is increased, the single-phase case shows higher wall temperatures (superheat) as compared to the boiling case, a pointer to the impact of boiling in reducing the wall temperatures.

Overall, these validations give some confidence in the Eulerian multiphase model implemented in FLUENT. Hence, we look at some IGBT package simulations involving boiling jets in the next section.

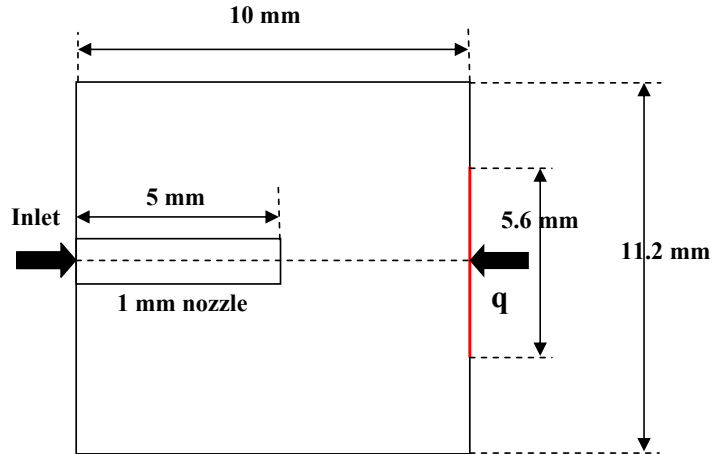


Figure 13. Domain for the Zhou and Ma (2004) study with submerged jets of R113 involving nucleate boiling

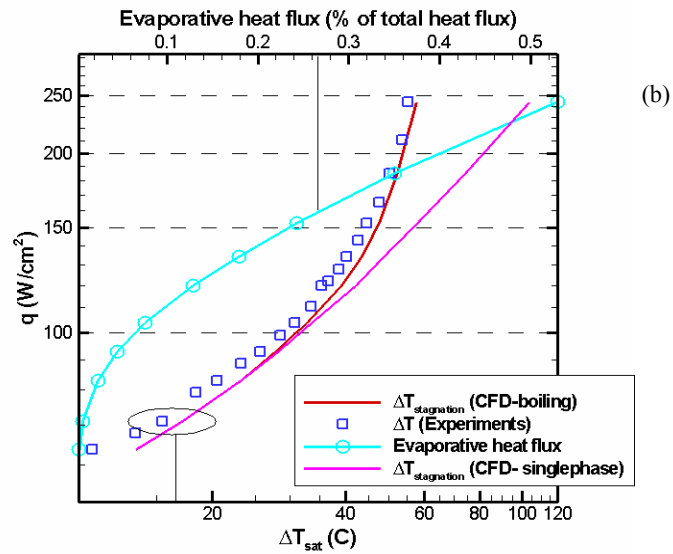
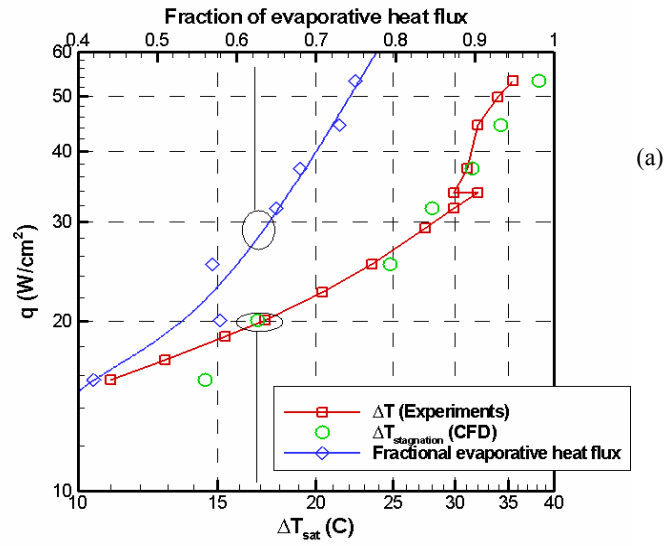
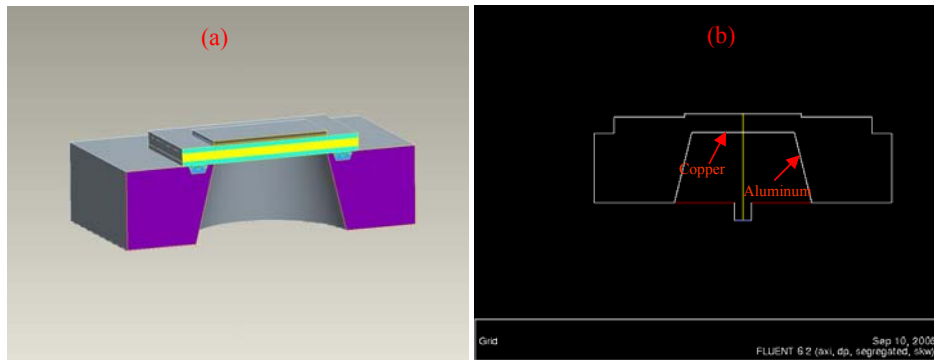


Figure 14. Boiling curve at different velocities for the Zhou and Ma (2004) study, (a) jet velocity of 0.41 m/s; (b) jet velocity of 11.36 m/s

IGBT package simulations with boiling jets. The previous section established some confidence in the CFD predictions of FLUENT by comparing them with experimental data from the literature. This section explores IGBT package simulations with boiling jets.

The axisymmetric domain used in the simulation is shown in Figure 15. The IGBT package consists of several layers. The silicon die is mounted on a direct bond copper (DBC) stack, which consists of an aluminum nitride layer sandwiched between two copper layers, and the DBC layer is attached to the aluminum baseplate/heat sink. Further details on the IGBT structure can be found in [14]. The boiling simulations are performed on a low resistance IGBT structure as shown in Figure 15(c). This low resistance IGBT structure is described in more detail in another milestone report that describes the development of a jet impingement cooling scheme for the IGBT package. These boiling simulations are fairly intensive and take several hours to converge on 64-bit Linux machines with 1.4 GHz processor speed, even with a small spatial mesh (i.e., small number of cells).

In the automotive industry, R134a is the working fluid used in the air-conditioning units. Here we explore the possible use of R134a as the cooling fluid for the IGBTs. The simulations presented here are performed with an R134a (properties given in Table 4) jet inlet temperature of 47°C. For the boiling simulations, the pressure is maintained at 1.32 MPa (i.e. ~ 13 atmospheres), at which the saturation temperature of the R134a is 50°C. So there is a 3°C sub-cooling in the R134a temperature at the inlet. A volumetric heat generation term is included in the silicon layer to simulate heat dissipation. The results for temperatures are mesh-independent to within 2%. The jet inlet velocity is 2 m/s (Figure 16(a)). Figure 16 shows the contours for velocity, vapor phase fraction, and temperature for the case of 75 W/cm² heat dissipation in the silicon die, a jet (diameter = 1.5mm) velocity of 2 m/s, a jet inlet temperature of 47°C, and with a saturation temperature of 50°C at a pressure of 1.32 MPa. Table 5 shows two sets of results—one case in which the heat flux dissipation in the silicon die is 75 W/cm² with the jet inlet velocity at 2 m/s, while another case in which the heat flux dissipation in the die is 200 W/cm² with the jet inlet velocity at 10 m/s. For both cases, jet inlet temperature is 47°C with a saturation temperature of 50°C (at 1.32MPa).



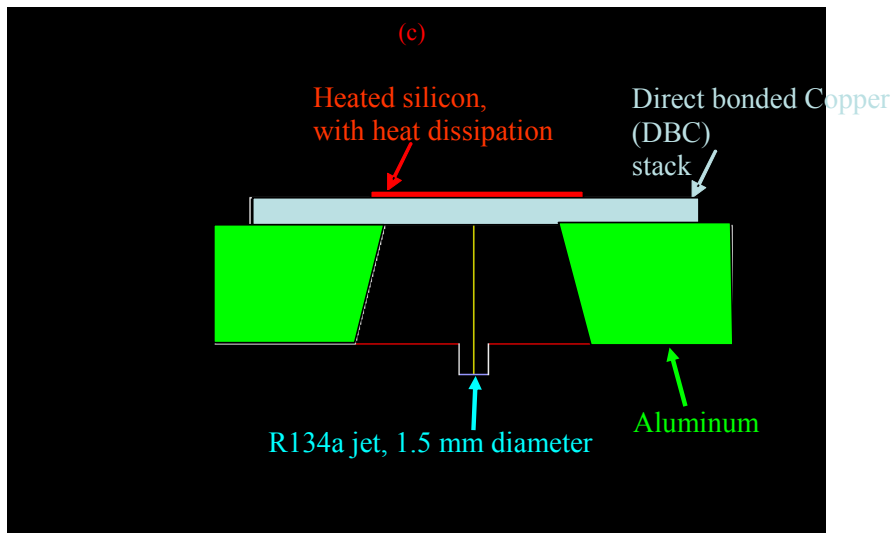


Figure 15. Axisymmetric domain used for the IGBT package simulation; (a) reduced thermal resistance IGBT structure; (b) axisymmetric domain used in the simulations; (c) different layers and some boundary conditions used

Table 4. Properties of R134a at 1.32 MPa, Saturation Temperature = 323.15K

	Liquid	Vapor
Density, kg/m ³	1103.4	66.16
Specific heat, J/kgK	1575	1219
Thermal conductivity, W/mK	0.0735	0.018
Dynamic viscosity, kg/ms	0.000167	0.0000138

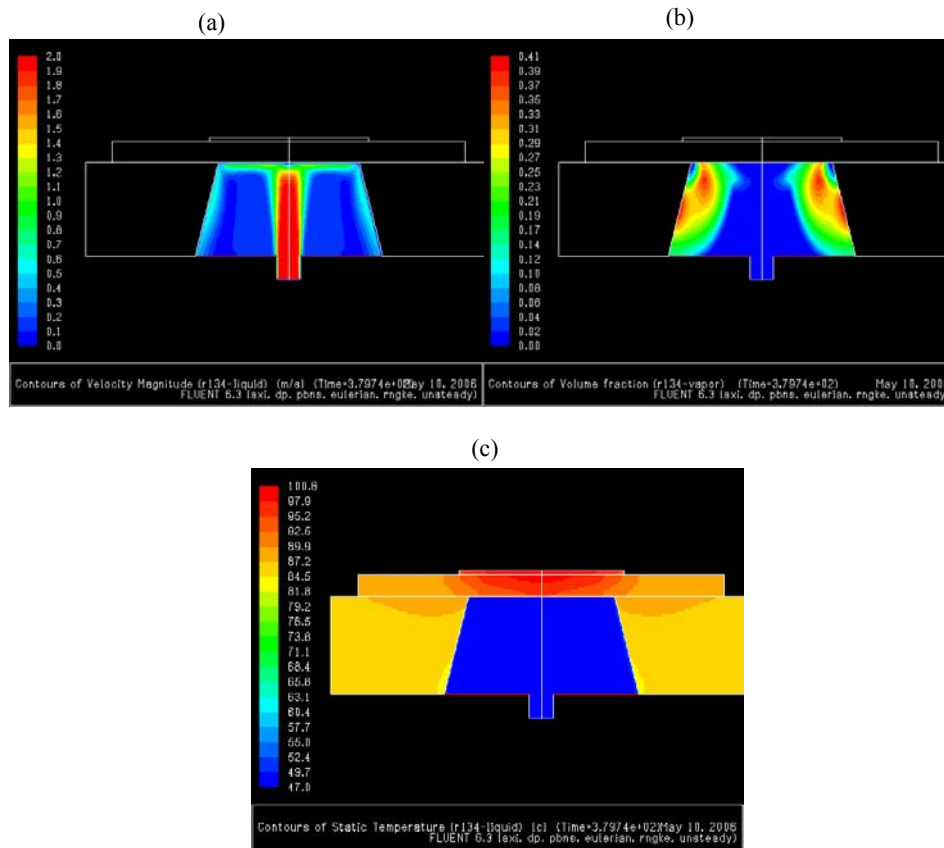


Figure 16. Representative contours of (a) velocity, (b) vapor fraction, (c) temperature in the domain for 75 W/cm², $T_{\text{inlet}} = 47^\circ\text{C}$, $T_{\text{sat}} = 50^\circ\text{C}$ at $P=1.32$ MPa

Table 5. Maximum Die Temperatures for R134a Jet (1.5mm diameter) Impingement Cooling of the IGBT Package Involving Nucleate Boiling, $T_{\text{inlet}} = 47^\circ\text{C}$, $T_{\text{sat}} = 50^\circ\text{C}$ at $P=1.32$ MPa

	75 W/cm ² , 2 m/s			200 W/cm ² , 10 m/s		
	T_{max} (°C)	h_{Cu} (W/m ² K)	h_{Al} (W/m ² K)	T_{max} (°C)	h_{Cu} (W/m ² K)	h_{Al} (W/m ² K)
Boiling	100.5	7,459	4,786	126.3	18,729	9,734
No boiling	115.4	6,088	3,300	110	26,476	14,321

Results are quite interesting. At a heat flux of 75 W/cm² and 2 m/s jet velocity, for the case involving boiling, the maximum die temperature is 100.5°C. For comparison purposes, a case is also shown where the boiling is “switched off”; i.e., it simulates single-phase flow. The temperature when there is no boiling is 115.4°C. This clearly demonstrates the impact of boiling in reducing the die temperature.

When the heat flux is increased to 200 W/cm² and the jet velocity is increased to 10 m/s, for the case involving boiling, the maximum die temperature is 126.3°C, while for the case when there is no boiling, the maximum temperature is only 110°C. This is an interesting result, and it suggests that boiling is not

beneficial at all jet velocities. The result here suggests that at elevated velocities, boiling may have a detrimental impact upon the heat transfer. We did an analysis that clearly shows why this happens. Basically, the vapor is associated with much smaller heat transfer coefficients than the liquid. So the appearance of vapor hinders heat transfer to some extent. It becomes a tradeoff between two opposing effects—the conversion to vapor aids in heat removal due to latent heat effects; however, the appearance of vapor also hinders heat transfer due to low heat transfer coefficients. These trends can be clearly seen in the overall averaged heat transfer coefficients (Table 5) at the aluminum and copper surfaces (the sides facing the fluid).

For the flux of 75 W/cm^2 and jet velocity of 2 m/s , the heat transfer coefficients for the boiling case are higher than for the case where there is no boiling. However, at a flux of 200 W/cm^2 and a velocity of 10 m/s , the heat transfer coefficients for the no-boiling case are higher than the case where there is boiling. Interestingly, this means that at some velocity and heat flux combination in between the two cases we have examined, the boiling and no boiling cases will yield roughly similar maximum die temperatures. From a broader FreedomCAR goal perspective, these simulations demonstrate conditions under which R134a can be used to come close to meeting the program goal of 200 W/cm^2 heat dissipation from the silicon die while maintaining the temperature close to 125°C .

These numerical results need to be validated experimentally. The experimental validations presented give some degree of confidence in the predictions from the code. It would be interesting to see how experimental results from the IGBT package involving boiling would match up with the predictions presented here. Overall, there is evidence of the benefits of boiling from a heat transfer standpoint – though it appears that boiling may not always be beneficial. Although experiments are essential, modeling can yield very useful information when used within their range of applicability. To the best of our knowledge, this is the first time such validations and package simulations are being reported. Tools such as those presented here can be explored for thermal design involving the use of R134a or any other fluid for thermal management in power electronics as well as other electronics cooling applications.

• **Two-Phase Sprays and Jets—Experimental Apparatus Construction**

In parallel with the numerical effort on two-phase jets, a subcontract was also established with Mudawar Thermal Systems in FY 2006 to fabricate a test vessel/experimental apparatus that would enable experiments with jets and sprays in the nucleate boiling regime. R134a and HFE7100/7200 have been identified as fluids which are particularly suitable for use as coolants for automotive applications. HFE7100/7200 can be used at atmospheric pressure; however, R134a has to be under pressure at about 15 atmospheres ($\sim 220 \text{ psi}$) in order for its saturation temperature to be at about 50°C – 60°C . Hence in order to conduct two-phase experiments with both R134a and HFE7100/7200, two different test vessels are being constructed. One of them can withstand high pressures (up to 600 psi) while the other can withstand up to 100 psi . The cross section of the test vessel and the experimental apparatus is shown in Figure 17. The test surface is a square of $10 \times 10 \text{ mm}$ area. The test surface is the top surface of an oxygen-free copper block. The copper block is surrounded by fiberglass insulation. Heat is supplied to it via cartridge heaters. One thermocouple is embedded into the copper block and the temperature of the center of the copper block close to the target surface is measured. The surface temperature is backed out from a conduction analysis. The liquid inlet temperature, the temperature in the chamber, the pressure in the chamber and the inlet pressure are all measured. A pressure relief valve is also provided. Figure 18(a) shows the assembled test vessel along with the instrumentation (close-up is shown in Figure 18(b)). The cartridge heaters and the oxygen free copper block along with the fiberglass insulation are shown in Figure 18(c). Figure 18(d) shows one sample spray nozzle from Spraying Systems Inc.

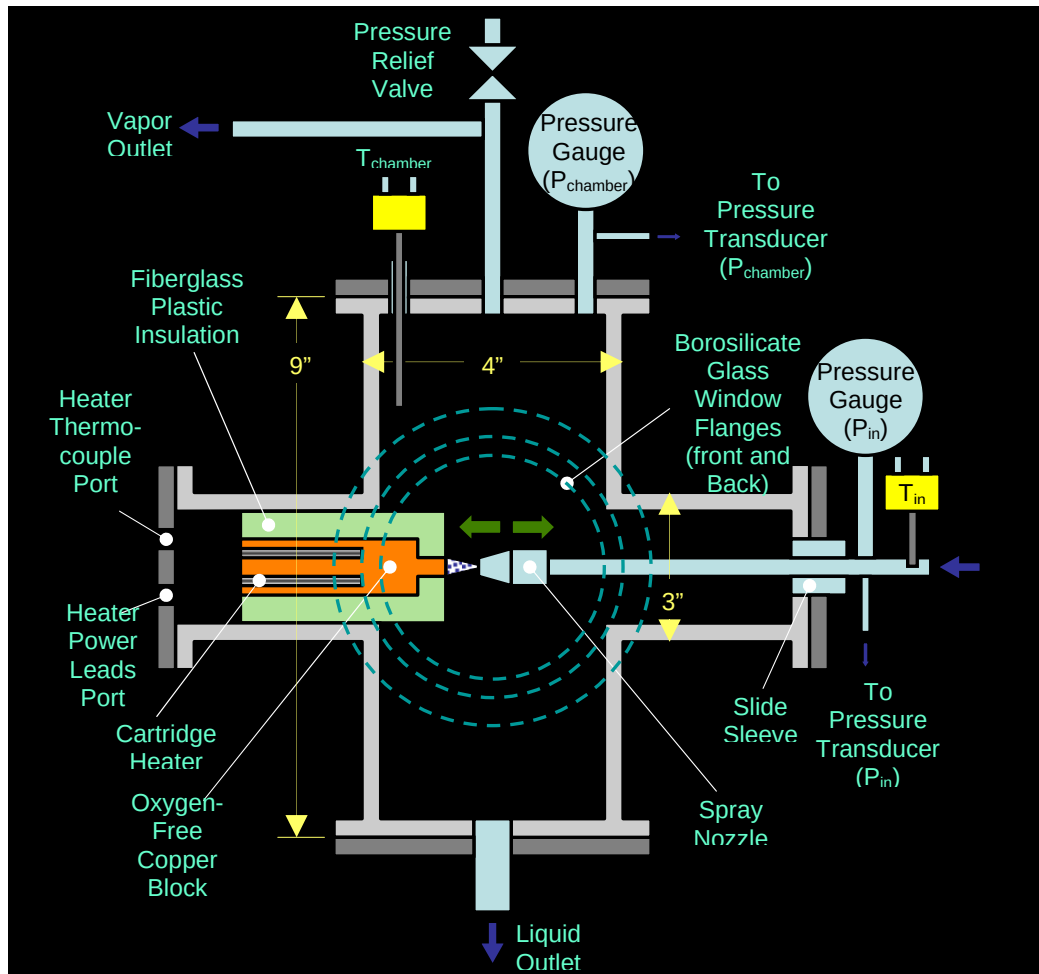
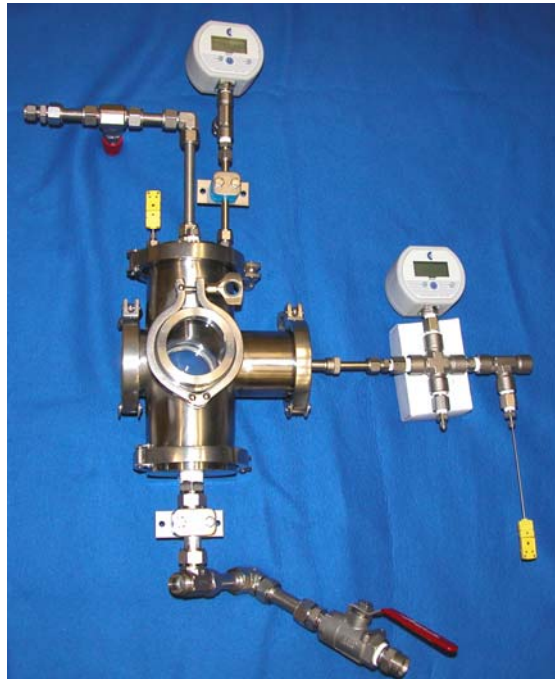


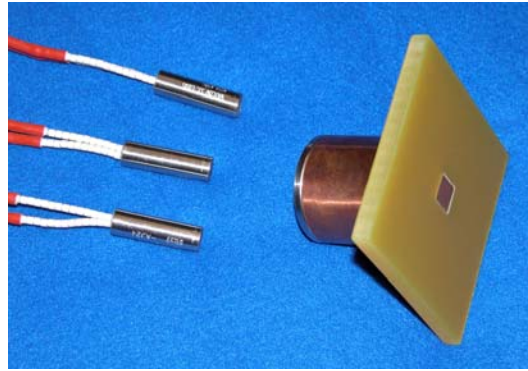
Figure 17. Cross-section of the experimental apparatus for two-phase spray/jet impingement cooling



(a)



(b)



(c)



(d)

Figure 18. Different components of the apparatus/test vessel, (a) test vessel with all the instrumentation; (b) close-up of the test vessel; (c) target surface surrounded by fiberglass insulation; cartridge heaters taken out and shown on the left; (d) spray nozzles from Spraying Systems Inc.

Based on what we have seen thus far in terms of single-phase and two-phase cooling techniques, a comparison between the two approaches is presented in Table 6. In order to achieve program goals with single-phase glycol water jets without using very high velocities, significant heat transfer and surface area enhancements would have to be obtained via concepts like self-oscillating jets and enhanced surfaces. The reliability of the system is yet to be established. With impinging jets, there are concerns related to erosion, corrosion and fouling of the surfaces on which the jet impinges. These aspects need to be quantified. The biggest advantage of using single-phase glycol-water jets is that no separate cooling loop would be required. With two-phase jets and sprays, the program goals can be met. However, it would require a separate cooling loop and additional components (e.g., condenser) would be required. For this case also, reliability is yet to be established.

Table 6. Comparison of Single-Phase and Boiling Jets/Sprays as Cooling Options

Single-phase jets (glycol-water)	- Existing cooling loop can be utilized. - With 105°C glycol-water coolant, to avoid very high velocities, heat transfer and surface area enhancements are required. - Reliability needs to be proven (erosion, corrosion, and fouling). - Program goals of 250 W/cm² and 125°C die temperature can be met with 70°C glycol-water coolant temperature with a heat transfer coefficient of $h=60,000$ W/m²K with the low resistance IGBT structure.
Boiling jets or sprays(with R134a or HFE7100/7200)	- A separate cooling loop is required - Some additional components are required (e.g. condenser). For R134a, a high pressure system is required - Reliability needs to be proven - Using R134a, conditions have been demonstrated numerically in this report for dissipating 200 W/cm² while keeping the die temperature at 125 C. HFE7100/7200 is also a promising coolant

Conclusions

Based on the results and discussion presented in the previous sections, it can be concluded that both single-phase jets as well as two-phase jets/sprays could potentially be used to meet the program goals of 250 W/cm² heat dissipation while maintaining the maximum die temperature of 125°C. Meeting the program goals with glycol-water jets at 105°C will be very challenging. Hence, there is a need to continue exploring concepts like self-oscillating jets as well as enhanced surfaces for improving the heat transfer rates as well as increase the surface area available for heat transfer. Initial results with self-oscillating jets show some promise; however, this technology will have to be thoroughly characterized. The problem is much more tractable with glycol-water jets at 70°C coolant temperature. Results presented in this report suggest that program goals can be met with heat transfer coefficients of the order of 60,000 W/m²K when 70°C glycol-water coolant is used. Conditions have been experimentally demonstrated in the report for achieving these heat transfer coefficients.

With two-phase jets/sprays, it is important to experimentally characterize the performance of fluids such as R134a, HFE7100/7200 and perhaps even glycol-water since there seems to be an interest in using glycol-water even in the two-phase regime. In this report, conditions have been demonstrated for dissipating 200 W/cm² while keeping the die temperature close to 125°C using jets of R134a. These numerical simulations show the promise that R134a holds in meeting program goals. HFE7100/7200 are also fluids which potentially could meet the goals. It will also be very critical to look at the system level implications for implementing these cooling technologies in an automotive system. It is important that system-level modeling be done to explore the feasibility of these cooling schemes from an overall systems perspective and also to understand practical aspects such as system complexity and cost.

Next Steps

The goal is to fully characterize the performance of the technologies that have been mentioned in this report; make this information available to industry; and guide the selection of the most promising technology considering performance, reliability, and cost.

Concepts such as single-phase self-oscillating jets to enhance heat transfer from the target surfaces will be fully explored. Surface enhancements may also be beneficial in increasing not only the surface area for heat transfer but also the heat transfer coefficients via changes in the flow. These enhancements may be required in order to meet the program goals with 105°C glycol-water coolant. In the coming year, we plan to shift focus on experiments with glycol-water at 105°C. Once the program goals are met on a test surface (say ½ inch diameter), experiments will be carried out on an actual IGBT package/inverter in order to prove these concepts.

Using glycol-water at 105°C and maintaining the silicon die temperature at 125°C is a challenging problem. As seen in earlier sections, even a heat transfer coefficient of 140,000 W/m²K on the DBC surface (without any surface enhancements) is not enough to keep the die temperature below 125°C. The problem becomes more tractable if the glycol-water coolant inlet temperature is 70°C.

Given that meeting program goals with 105°C glycol-water coolant may be very challenging, two-phase jet/spray impingement cooling also is being explored. Considerable numerical work has been performed in FY 2005 and FY 2006 relating to nucleate boiling in impinging jets. Hence, in FY 2007, experiments will be performed to characterize the performance of two-phase jets and sprays of coolants such as R134a, HFE7100/7200 and perhaps even glycol-water in the two-phase regime.

Some other aspects that are of concern are erosion, corrosion, and fouling which can all be categorized under issues related to reliability. It is not entirely clear at this point what would be the best cooling option from an overall systems context. Hence, the immediate goal is to characterize the heat transfer performance of these different cooling options and then incorporate all these options into an overall systems level modeling framework.

References

- [1] Zumbrennen, D.A. and Aziz, M., 1993, "Convective heat transfer enhancement due to intermittency in an impinging jet," *ASME Journal of Heat Transfer*, **115**, pp. 91–98.
- [2] Mladin, E.C. and Zumbrennen, D.A., 1994, "Nonlinear dynamics of laminar boundary layers in pulsatile stagnation flows," *Journal of Thermophysics and Heat Transfer*, **8**(3), pp. 514–523.
- [3] Mladin, E.C. and Zumbrennen, D.A., 1997, "Local convective heat transfer to submerged pulsating jets," *International Journal of Heat and Mass Transfer*, **40**(14), pp. 3305–3321.
- [4] Camci, C. and Herr, F., 2002, "Forced convection heat transfer enhancement using a self-oscillating impinging planar jet," *ASME Journal of Heat Transfer*, **124**, pp. 770–782.
- [5] Narumanchi, S.V.J., Amon, C.H. and Murthy, J.Y., 2003, "Influence of pulsating submerged liquid jets on chip-level thermal phenomena," *ASME Journal of Electronic Packaging*, **125**, pp. 354–361.
- [6] Narumanchi, S.V.J., Troshko, A., Hassani, V. and Bharathan, D., 2006, "Numerical simulations of boiling jet impingement cooling in power electronics," *ITherm conference*, San Diego, CA.
- [7] Narumanchi, S.V.J., Troshko, A., Bharathan, D. and Hassani, V., 2006, "Numerical simulations of nucleate boiling in impinging jets: applications in power electronics cooling," *Int. J. Heat Mass Transfer*, **to be submitted**.
- [8] Wolf, D.H., Incropera, F.P. and Viskanta, R., 1993, "Jet impingement boiling, *Advances in Heat Transfer*, **23**, pp. 1–132.
- [9] Garimella, S.V., 2000, "Heat Transfer and flow fields in confined jet impingement," in *Annual Review of Heat Transfer*, pp. 413–494.
- [10] Lienhard, J.H., 1995, "Liquid jet impingement," in *Annual Review of Heat Transfer*, pp. 199–270.

- [11] Webb, B.W. and Ma, C.-F., 1995, "Single-phase liquid jet impingement heat transfer," *Advances in Heat Transfer*, **26**, pp. 105–217.
- [12] Womac, D.J., Ramadhyani, S. and Incropera, F.P., 1993, "Correlating equations for impingement cooling of small heat sources with single circular liquid jets," *ASME Journal of Heat Transfer*, **115**, pp. 106–115.
- [13] Garimella, S.V. and Rice, R.A., 1995, "Confined and submerged liquid jet impingement heat transfer," *ASME Journal of Heat Transfer*, **117**, pp. 871–877.
- [14] Narumanchi, S.V.J., Hassani, V. and Bharathan, D., 2005, "Modeling single-phase and boiling liquid jet impingement cooling in power electronics," National Renewable Energy Laboratory, NREL Report No. TP-540-38787. www.nrel.gov/docs/fy06osti/38787.pdf.
- [15] Katto, Y. and Kunihiro, M., 1973, "Study of the mechanism of burn-out in boiling system of high burn-out heat flux," *Bull. JSME*, **16**(99), pp. 1357–1366.
- [16] Zhou, D.W. and Ma, C.F., 2004, "Local jet impingement boiling heat transfer with R113," *Heat and Mass Transfer*, **40**, pp. 539–549.

Publications

The following are the publications and presentations for FY 2006.

Publications

1. Narumanchi, S.V.J.; Troshko, A.; Hassani, V.; and Bharathan, D. 2006, "Numerical simulations of boiling jet impingement cooling in power electronics," *Proceedings of the 2006 ITherm conference*, May 29-June 2, San Diego, CA.
2. Narumanchi, S.V.J.; Troshko, A.; Hassani, V.; and Bharathan, D. 2006, "Numerical simulations of nucleate boiling in impinging jets: applications in power electronics cooling," to be submitted to the *International Journal of Heat and Mass Transfer*.

Presentations

1. Narumanchi, S.V.J.; Troshko, A. 2006, "Numerical simulations of boiling jet impingement cooling in power electronics," *CFD User's Summit*, Monterrey, CA, May 22–24, 2006.

Nomenclature

C_p	Specific heat, J/KgK
d	Jet diameter, m
D	Target equivalent diameter, m
DBC	Direct bonded copper (Copper, aluminium nitride and copper layers bonded together)
f	Frequency, Hz
$\vec{g}$	Gravitational acceleration vector, m/s ²
h	Heat transfer coefficient, W/m ² K
IGBT	Insulated-gate bipolar transistor
k	Thermal conductivity, W/mK
p	Pressure, N/m ²
Pr	Prandtl number
q, q''	Heat flux, W/m ²
Re	Reynolds number
u, U	Liquid velocity, m/s

v	Phase velocity, m/s
	Jet velocity at the nozzle exit, m/s

Greek symbols

ΔT	Temperature difference, K
μ	Dynamic viscosity, Ns/m ²
ν	Kinematic viscosity, m ² /s
ρ	Density, Kg/m ³
σ	Surface tension, N/m

Subscripts

avg	Average
CHF	Corresponding to critical heat flux
d	Jet diameter
eff	Effective
f	Corresponding to fluid
l	Corresponding to liquid
q	Corresponding to phase q
sat	Corresponding to saturation
sub	Corresponding to subcooling

2.5 Low Thermal Resistance IGBT Structure

Principal Investigator: Keith Gawlik

National Renewable Energy Laboratory

1617 Cole Boulevard

Golden, CO 80401

Voice: 303-384-7515; Fax: 303-384-7540; E-mail: keith_gawlik@nrel.gov

DOE Technology Development Manager: Susan A. Rogers

Voice: 202-586-8997; Fax: 202-586-1600; E-mail: Susan.Rogers@ee.doe.gov

Abstract

This report summarizes the FY 2006 activities in reducing thermal resistance between the silicon die of the IGBT and the coolant. Activities included simulation and analysis of enhanced IGBT structures, jet impingement experimental studies, and the development of prototype parts. From the jet impingement studies, we were able to conclude that heat transfer increases as jet impingement increases and heat transfer increases as the nozzle-to-target spacing is reduced. Simulation and experiment resulted in a design that allows 105°C coolant at 100 W/cm² heat flux and keeps die temperatures at acceptable levels. The structural, fluid, and thermal simulations performed on various geometries for the nozzle plate and base plate have resulted in a design ready for prototype evaluation. Experimental testing and optimization of the design is planned for the coming year.

Objective

The objective of this project is to reduce the thermal resistance between the silicon die of the insulated gate bipolar transistor (IGBT) and the coolant. By reducing the thermal resistance, a target coolant temperature of 105°C can be used to cool the inverter electronics. This project will address one of the important targets of the Advanced Power Electronic and Electric Machines (APEEM) Program, which is the use of coolant at 105°C.

Approach

In previous fiscal years, NREL proposed eliminating several layers in the structure of the IGBT module to assist its effective cooling by a 105°C inlet coolant temperature. In the proposed layout, holes of appropriate dimension are cut through the base plate all the way to the direct bonded copper (DBC) layer of the IGBT. Nozzles are positioned under the DBC direct coolant onto the target surface in high-speed jets. The jets are oriented such that they are directly under the IGBT and diode dies. Therefore, the coolant comes into direct contact with the DBC layer and eliminates the thermal resistances of the base plate and the thermal interface material; the latter causes about 40% to 50% of the total thermal resistance of the IGBT structure. NREL created several thermal and structural models of the proposed low thermal resistance architecture. The numerical modeling shows very promising results and the thermal resistance can be reduced by as much 60%. The weight of the base plate can be reduced by as much as 51% because portions of the base plate are eliminated to allow direct coolant access to the DBC. This structure has

been modeled with various size holes, and structural and thermal analyses have been completed. Currently NREL is conducting fatigue and thermal cycling analysis on the proposed structure.

The idea of a low thermal resistance IGBT structure has been extensively examined with ANSYS, CFX, FLUENT, and other thermal and structural modeling tools. In addition, the actual hardware testing and evaluation are needed for proof of concept. Therefore, NREL has also designed and built small- and full-scale prototypes to validate the modeling results observed through simulation.

This project is strongly aligned with DOE's FreedomCAR objectives. It addresses a major issue facing the industry: the use of coolant at a 105°C inlet temperature. Our analysis shows that a temperature difference as low as 21°C for a heat flux of 90 W/cm² is feasible. This is far more than what the current technology can do, which is to remove about 60 W/cm² at a temperature difference of about 50°C.

In addition to continuing the refinement of the IGBT nozzle and plenum designs through simulation, in this fiscal year we prepared a simple, small scale, six-jet low thermal resistance IGBT module for laboratory testing and evaluation. We constructed and tested a set of IGBTs with the low thermal resistance structure in the laboratory environment with a die temperature of 125°C and the inlet coolant temperature of 105°C. A water-glycol mixture was used as the coolant and the back of the DBC was cooled by the single phase jet impingement technique. The tests simulated the structure of the Semikron inverter that has been used in our modeling activities.

At the end of this fiscal year NREL worked with Semikron through a CRADA, to build a prototype of the proposed structure in an actual inverter. Prototype nozzle and base plate parts have been machined, and the parts will be sent to Semikron to add its module. Next fiscal year, the whole unit will be tested for various operating conditions and proof of concept.

If successful in long-term testing, the new low thermal resistance design will have a significant impact on the design of the inverter unit. It will allow the industry to eliminate a separate cooling loop and allow use of the coolant exiting the engine at 105°C to cool the power electronics unit directly. It will also reduce the weight of the heat sink by as much as 51%. The project will help DOE achieve one of its major targets of using 105°C coolant inlet temperature for cooling of power electronics. The targets for thermal management of power electronics and electric machines are not firmly set yet. However, based on the current cold plate design, the heat sink constitutes 32% of the weight of a Semikron inverter. This project will reduce that fraction to 16% of the total weight of the inverter. The current technology uses 75°C coolant inlet temperature. This project will meet the target of 105°C inlet coolant temperature. The current technology can remove 60 W/cm² heat flux; the proposed structure will remove 90 W/cm², or 50% more.

Major Accomplishments

Detailed structural and thermal analyses and experimental testing of conventional and low thermal resistance IGBT structures have been performed with the finite element and finite volume modeling software packages ANSYS, CFX, and FLUENT, and show a reduction in thermal resistance of at least 64%. These results have led to a patent being filed for the cut-through IGBT structure invention, while modeling design refinement continues as subtleties in the flow and thermal fields are understood. Results also indicate that the high heat transfer available through jet impingement cooling can limit the maximum operating temperature of the dies to 125°C with an inlet temperature of 105°C given power densities as high as 100 W/cm². This is a 66% higher heat transfer rate than a conventional IGBT structure can handle with 75°C coolant.

- Modeling of next-generation low thermal resistance architecture is complete.
- Experimental evaluation of six-jet structure is complete.
- Results indicate 64% less thermal resistance than the conventional configuration.
- Results indicate viability of 105°C inlet coolant temperature.
- Prototype parts have been machined and will be sent to Semikron for assembly of a complete, functioning unit.

Technical Discussion

Simulation and analysis of enhanced IGBT structures

Implementing this new concept required redesigning the cooling plate to maximize the area exposed to the impinging jets and designing the hardware required to create the jets. The basic concept of this design can be seen in Figure 1 in an exploded format. The aluminum base plate, nozzle plate, and flow channel plate have been designed as individual components to reduce the cost of fabrication. Each component can be directly cast with minimal machine finishing to provide an inexpensive end product to the consumer.

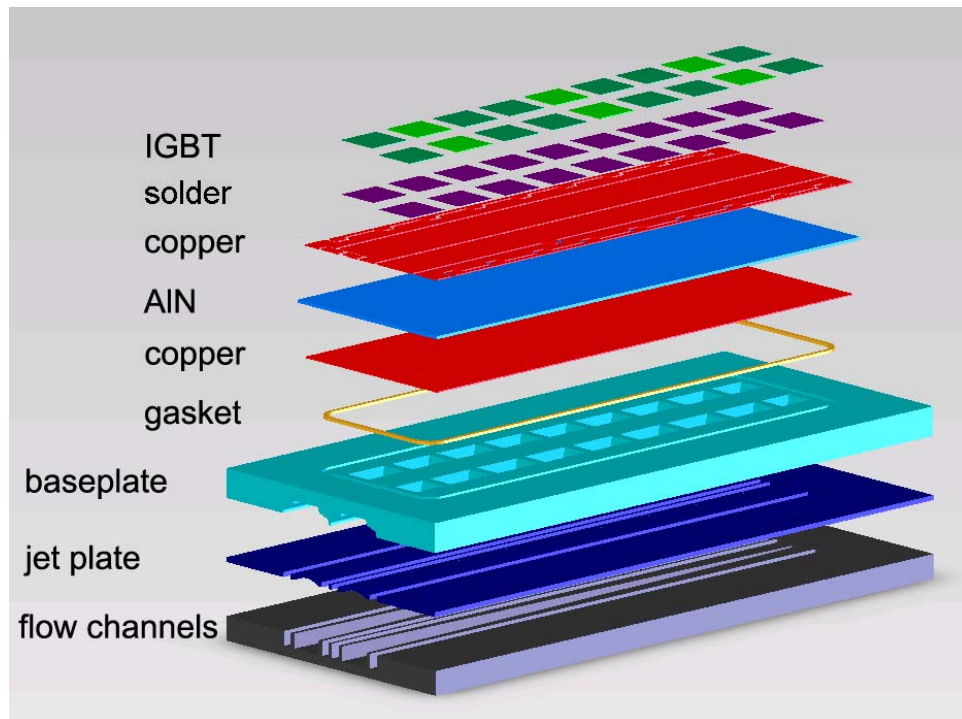


Figure 1. Exploded view of the low thermal resistance IGBT structure

The cut-through design of the aluminum cooling plate for the IGBT structure is an effective solution to the problem of minimizing the thermal resistance to provide maximum cooling. In this layout, the 18 holes lie directly below the respective power electronic dies. In the latest generation of the design, shown in Figures 2 and 3, the individual holes have been merged into open rows, with a bridge between the rows that help distribute the force applied to the electrical contacts. These bridges help to support the phase of the inverter from excessive bending and reduce the risk of cracking. The cut-through allows the jets of coolant fluid to impinge directly on the DBC to which the IGBTs and power diodes are soldered. The benefits of this design are the lack of thermal grease and the ability to have high heat transfer coefficients on the DBC target surfaces.

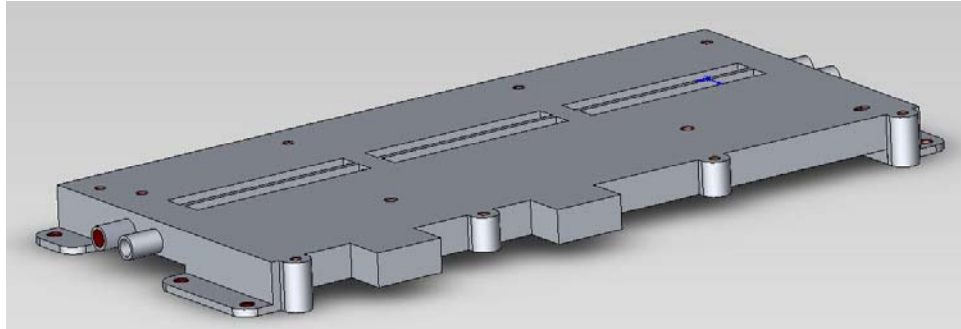


Figure 2. Base plate of the latest design, showing the elongated open holes under the rows of IGBTs and diodes

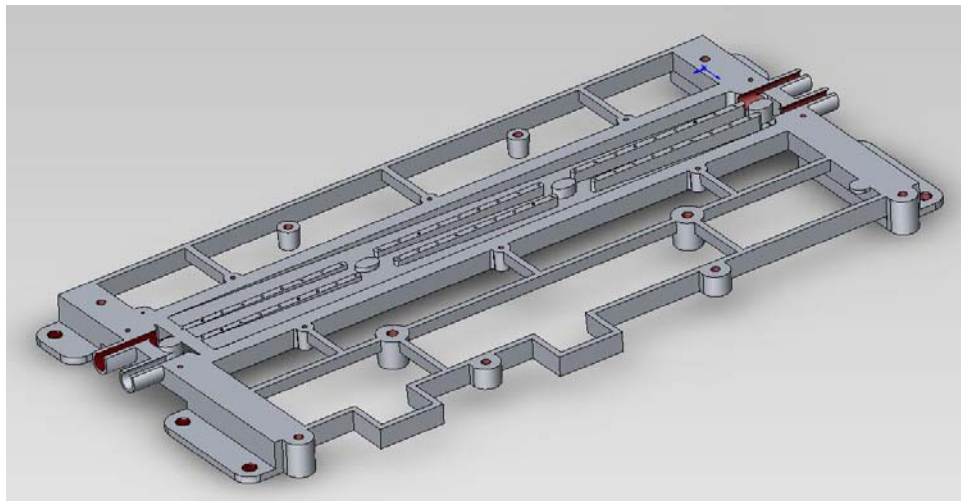


Figure 3. Cutaway view of the nozzle plate located under the DBC

The elongated holes are surrounded by a groove for an O-ring to seal the DBC and the aluminum. An O-ring must be in place to prevent the coolant from penetrating into the electronics side. Using an O-ring allowed us to eliminate any rigid bonding between the heat sink and the DBC. This allows the two surfaces to freely expand and contract as the temperature varies, which decouples the strain and reduces the thermally induced stress. Figure 4 shows the top view of the circuit board, which contains the IGBTs

and diodes, and which is mounted on the elongated holes of the base plate. The opposite side is sealed to the base plate with O-rings.

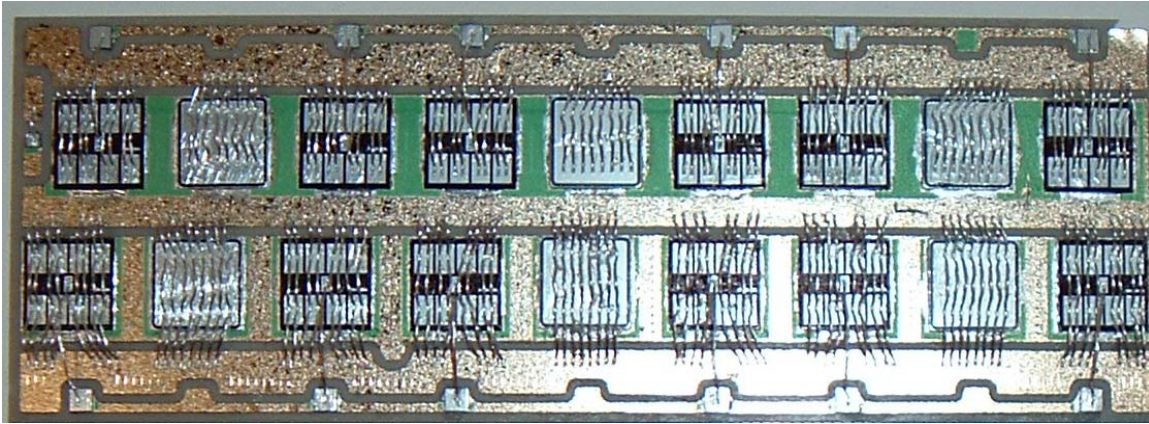


Figure 4. Top view of the circuit board containing IGBTs and diodes, which are mounted to the base plate. The DBC surface on the opposite side is cooled by the impinging jets.

Simulations using structural, heat transfer, and fluid dynamics finite element and finite volume codes have resulted in an advanced design ready for prototyping. Maximum stresses and deflections (Figure 5) were found in the structural simulations of the latest design to be acceptable with Semikron's assembly process, which applies 3000 psi to the DBC assembly. In the thermal and fluid simulations, heat flux was set to 100 W/cm^2 , for a total heat load of 3.6 kW. The coolant flow rate was 20.4 lpm. Jet heat transfer coefficients derived from Narumanchi's numerical work showed the maximum temperature to be 125.8°C with a 105°C inlet temperature, as shown in Figure 6. The results showed only a 4°C temperature difference between the DBC target surface and jet inlet temperature.

Fluid flow simulations were performed to obtain uniform flow from all jets (Figure 7) and to explore ways to minimize jet-to-jet interactions and smoothly exhaust spent coolant. Maximum jet speed was 10.6 m/s at the given flow rate and with 1.5-mm diameter nozzles. When the jets are submerged, as is anticipated in actual operation, the impingement velocity at the target decreases somewhat compared to a free surface jet, but the heat transfer coefficient is not adversely affected (Figure 8). The average heat transfer coefficient over the cut-through region is $20,560 \text{ W/m}^2\text{K}$.

Jet interaction does not substantially decrease the heat transfer coefficient. Pressure drop through the assembly is 7.5 psi, which compares favorably to Semikron's current elliptical fin cold plate design.

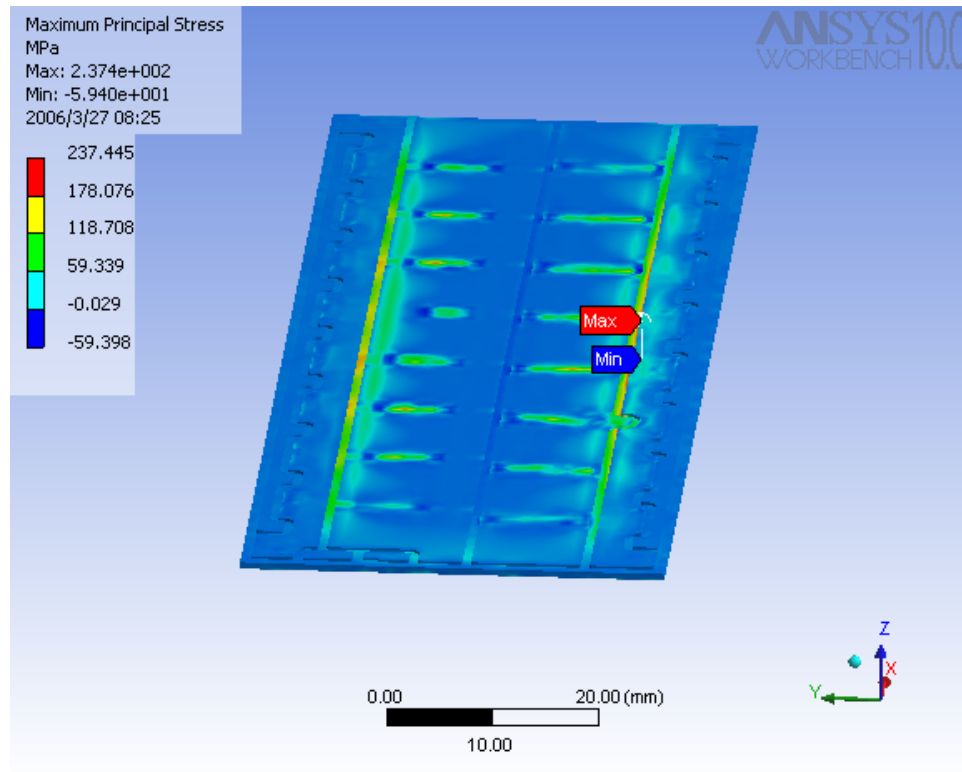


Figure 5. Maximum stresses in the assembly were acceptable to Semikron

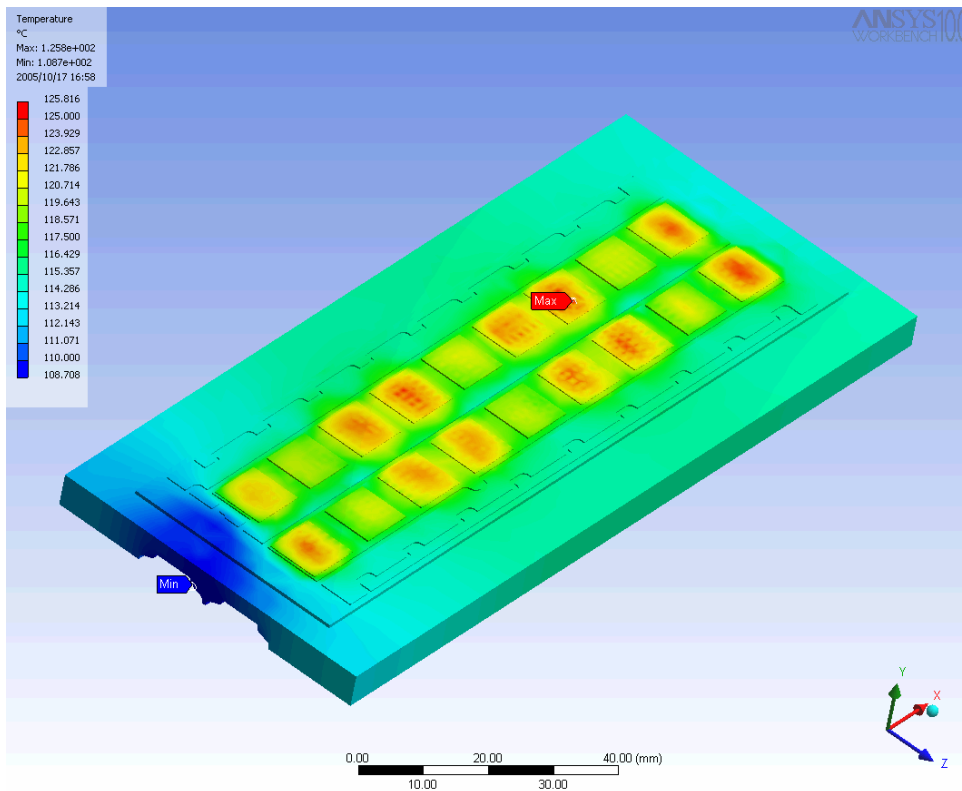


Figure 6. Die temperature contours at 100 W/cm^2 , 105°C inlet temperature, 20.4 lpm flow rate

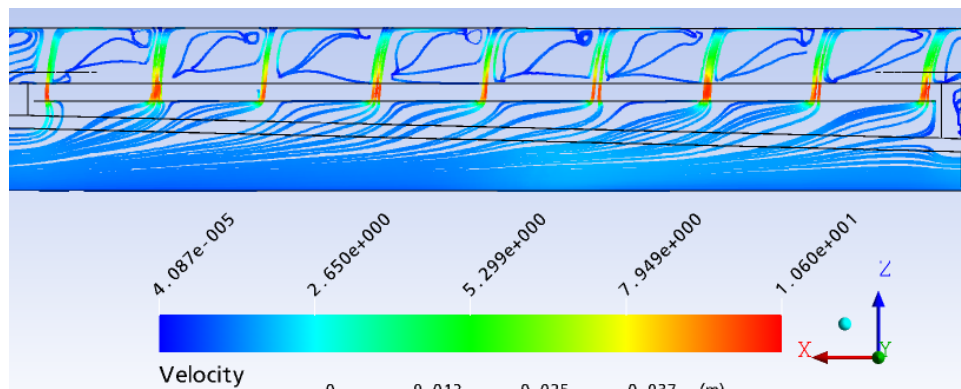


Figure 7. Uniform jet flow is obtained from the latest generation nozzle plate

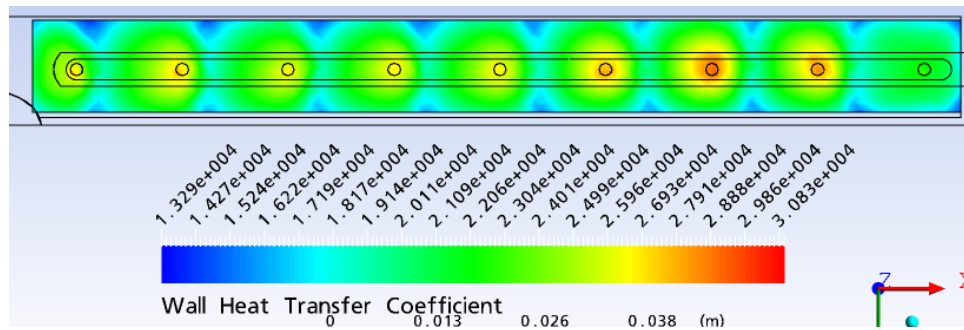


Figure 8. Heat transfer coefficients on target surface, with outline of nozzle plate shown below

Jet impingement experimental study

Jet impingement heat transfer has been studied and documented for its proven ability to attain high heat transfer coefficients. The primary focus of previous studies has been in three classes of impinging jets: submerged, confined, and free surface. Many times in practice, the available topology of the surfaces do not lend themselves to one of these configurations. Structural and other geometric considerations often make the impingement of jets into cavities an attractive solution for those wishing to optimize heat transfer performance with impinging jets.

As power requirements continue to increase in the realm of power electronics, the ability to extract the maximum heat flux is desirable. For that reason, an experimental study was performed on the geometric influences of impinging jets in cavities. This parametric study determined the effect of four variables on the average heat transfer coefficient on a constant given area: nozzle-to-target distance, velocity, chamfer angle, and geometric cavity shape.

Single-phase liquid jets have been studied extensively in the literature, which includes experiments, theoretical analyses, and numerical simulations. Different configurations of impinging jets, including single free-surface jets, multiple free-surface jets, single submerged jets, multiple submerged jets, and confined single submerged jets, have been studied. Planar and circular jets have also been studied. In the field of electronics cooling, many experimental correlations have been developed for the local and average heat transfer coefficients on the surface of the simulated chip. Most of the simulated chips are either 10×10 mm or 12.7×12.7 mm. Air jets have also been studied extensively. Some of the nondimensional heat transfer correlations developed from studies on air jets can be applied to liquid jets. Researchers have explored the impact of a vast array of parameters such as jet velocity, jet diameter, impact angle, nozzle-to-nozzle spacing, turbulence level, nozzle shape, nozzle length, jet pulsations, jet confinement, chip-surface enhancement, and fluid properties on the chip surface heat transfer coefficients. All these are covered in detail in comprehensive reviews.

In hybrid electric cars, inverters perform the DC/AC conversion. The inverters contain a number of IGBTs, which function as on/off switches. A substantial amount of power is dissipated by the IGBTs. Each IGBT has a silicon die area of 9×9 mm, with a thickness of 0.25 mm. One goal of this part of the project was to explore and compare the single-phase jet impingement configurations reported in the literature for cooling IGBTs. Typically, the package consists of the silicon die, copper layer, aluminum nitride, copper layer, aluminum base plate, and heat sink. In automotive applications, a 50:50 mixture of ethylene glycol and water is typically used as a coolant. In this project, we first considered a chip of 10×10 mm and looked at the heat transfer performance of single-phase jet impingement configurations such as single and multiple free-surface jets, single and multiple submerged jets, and confined jets. We restricted ourselves to circular jets at this point. Different empirical correlations from the literature were

used to gain insights into the average heat transfer coefficients that can be obtained from the chip surface. Numerical simulations (computational fluid dynamics) were performed for some of the configurations reported in the literature and were used in the simulations described in the preceding section. Simple calculations were also performed to understand the pressure drop associated with some of the jet impingement configurations.

Another practical concern associated with these systems is erosion. High-velocity jets (>5 m/s) have the potential to erode the material onto which they impinge.

IGBT package simulations were also performed with a glycol-water mixture and water as fluids. The jet inlet velocity temperature was maintained at 105°C , and simulations were performed with two heat fluxes of 90 and 200 W/cm^2 .

The apparatus used for this experiment comprised several parts and tested two concepts. The first was a simple test to verify that a standard double-seal O-ring could adequately seal the inner chamber from the environment at pressures as high as 100 psi and at elevated temperatures. The second was to verify and determine the effects of flow rate and nozzle-to-target spacing on heat transfer. The first objective was accomplished by completely assembling the apparatus and using compressed air at 150 psi to test for leaks. The system was pressurized and the supply of air was removed while the pressure within the test vessel was monitored. Stable pressure indicated that the assembly was free from leaks and the O-ring provided an adequate seal.

The next focus of the experiment was to determine the effects of nozzle spacing and jet velocity on heat transfer. The experiments were carefully designed and conducted to extract the most information from the data. The apparatus contained six resistors to provide heat fluxes up to 140 W/cm^2 , adjustable nozzles and flow rate, and heat transfer coefficients up to $35,000\text{ W/m}^2\text{K}$. Figure 9 shows the array of resistors mounted on the apparatus. Figure 10 shows the apparatus opened up, with the target surfaces under the resistors displayed. Figure 11 shows the set of six nozzles, which can be moved closer or further away from the target surfaces. Figure 11 also shows the O-ring groove.

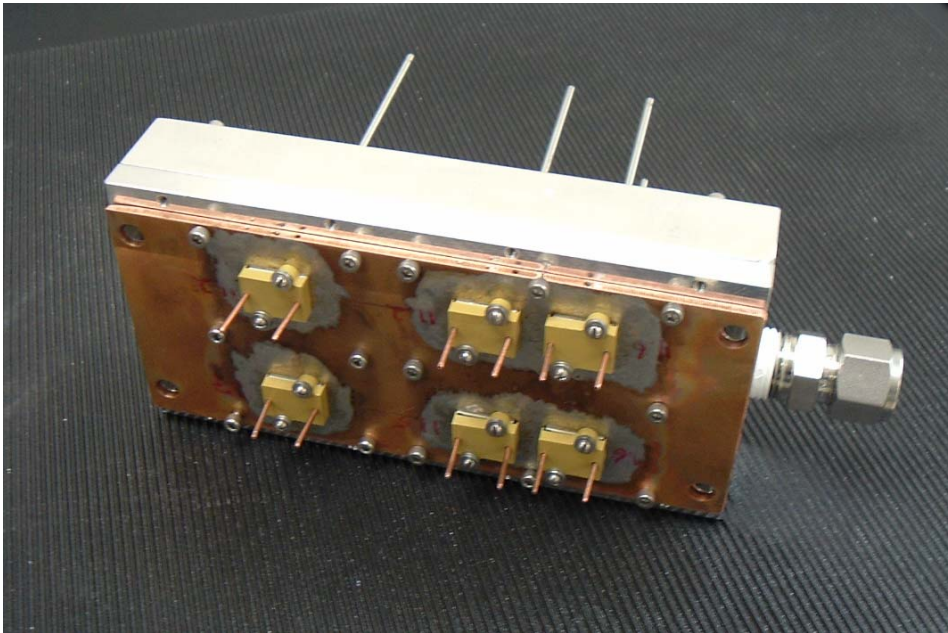


Figure 9. Resistors mounted on experimental apparatus

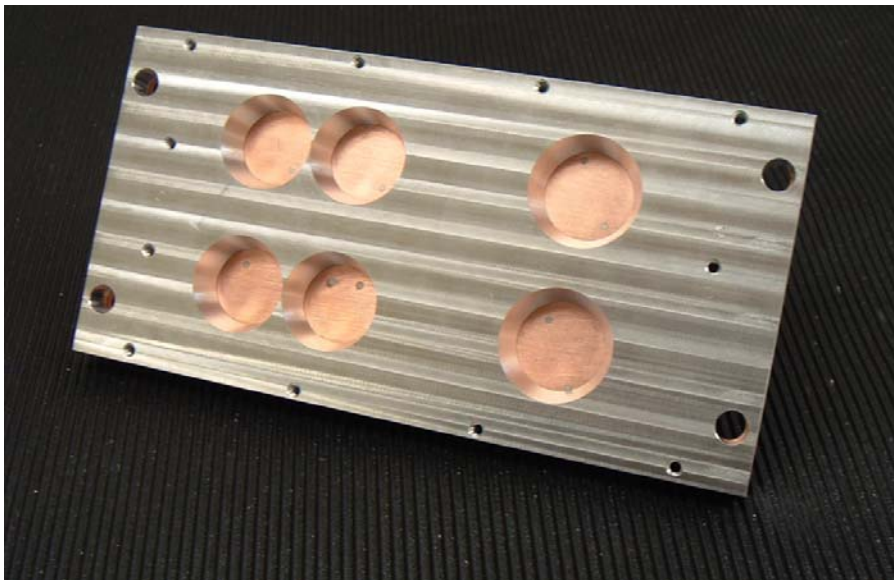


Figure 10. View of target surfaces, under resistors

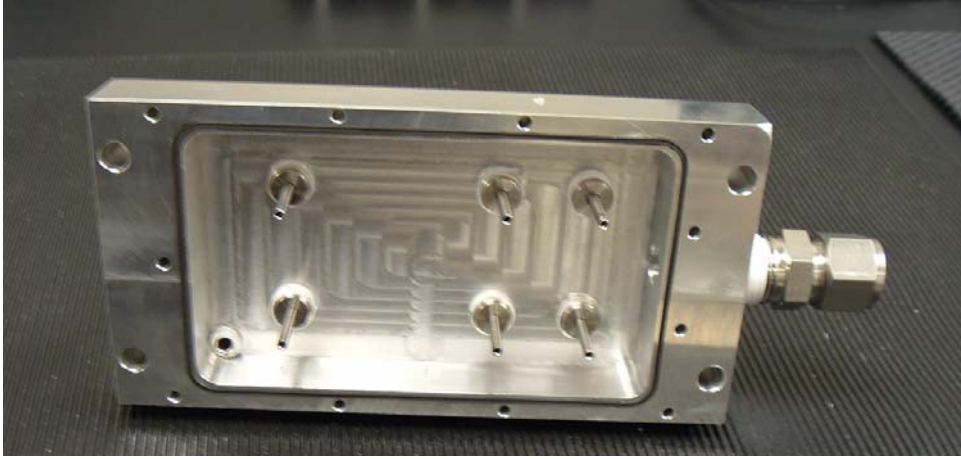


Figure 11. Nozzle array

Matlab's Model Based Calibration toolbox was used to analyze the data collected from the series of experiments. The domain space was mapped onto a homomorphic region through the use of coded values to build a response model and corresponding response surface. Validations of the model and subsequent error distribution were further analyzed.

After evaluating several models for response surfaces, including radial basis functions and polynomials up to fifth order, we chose a third order polynomial with a corresponding $R^2 = 0.9956$.

The coded values were then used in the polynomial expansion of the response surface. The resulting equation for the difference in temperature, DT , as a function of the coded nozzle to target spacing, x_1 , and the flow rate, x_2 , is

$$DT = 9.0579 - 0.65132x_1^2 - 1.7798x_2^2 + 1.0206x_1^2 - 1.2349x_1x_2 + 0.90627x_2 + 1.3586x_1^3 + 1.8056x_1^2x_2 - 2.1525x_1x_2^2 - 1.939x_2^3$$

A simple sensitivity analysis of the coefficients in the equation confirmed the high degree of interactions between the two parameters in this study and is confirmed by the convoluted topology exhibited in the response surface.

The use of jet impingement heat transfer has demonstrated that heat transfer can be vastly improved by manipulating the nozzle-to-target spacing of the nozzle and by adjusting the flow rate and consequent velocity of the jet. By constructing and sampling a robust design space, we could analyze the data collected and propose a model. The model described by the equation when compared to the experimental data had a relevance of $R^2 = 0.9956$. Further analysis, validation, and error estimates of the data and resulting model showed that for the data collected and the sampling space, the model is justifiably robust. Error is minimized in regions except where the velocity is extremely high and the nozzle is extremely close to the target surface.

The general trends exhibited in the data are primarily as follows:

- Heat transfer increases as the velocity increases.

- Heat transfer increases as the nozzle-to-target spacing is reduced.

These two generalizations certainly have exceptions, but generally hold true. The equation can be used to optimize jet spacing and velocity given identical geometric layouts specified in this experiment.

Prototype parts

The simulation and experimental work resulted in Generation 13 of the design being chosen as the best one to prototype. The parts were machined in August 2006 and will be sent to Semikron for the addition of the module. Figure 12 shows the base plate and nozzle plate parts. The base plate is shown with the bottom side up. This side will receive the nozzle plate and will be sealed with a cover plate. The inlet and outlet fluid ports are seen on either side of the base plate. The nozzle plate is shown with the jet side up. Figure 13 shows the nozzle plate put into position in the base plate. Figure 14 shows the base plate with the module mounting surface up. The elongated holes, over which the DBC (with the IGBTs and diodes) is mounted, are shown. The nozzles can be seen through the elongated holes. The bridges running between the holes support the assembly. Figure 15 shows a close view of the nozzles, holes, and O-ring grooves. The nozzles are placed directly under the IGBTs and diodes, which are not spaced evenly. The O-ring groove is designed to accept a quad-style ring, which was found in the experimental work to seal the assembly up to 150 psi. Figure 16 is another view of the nozzle plate, showing how the nozzles are placed according to the IGBT and diode locations, and the angled plenums on either side of the plate, which promote uniform flow from each nozzle. Figure 17 shows one end of the nozzle plate in a close view.

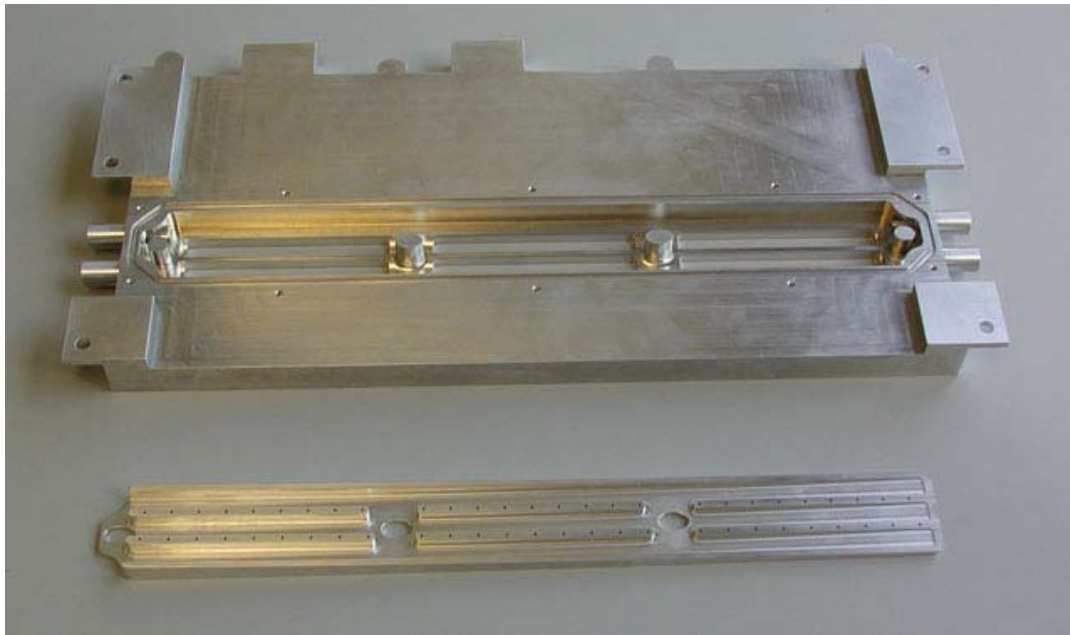


Figure 12. Base plate and nozzle plate parts

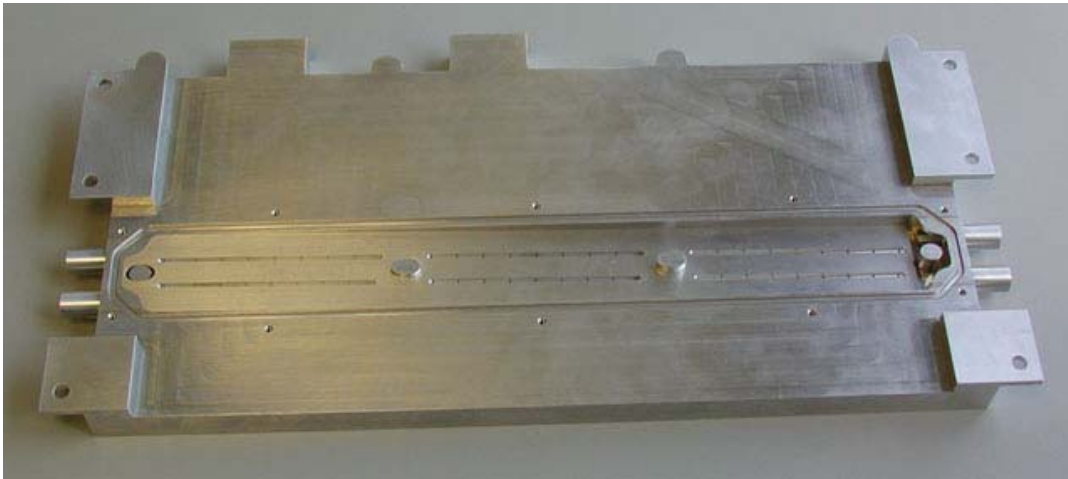


Figure 13. Nozzle plate in position in the base plate

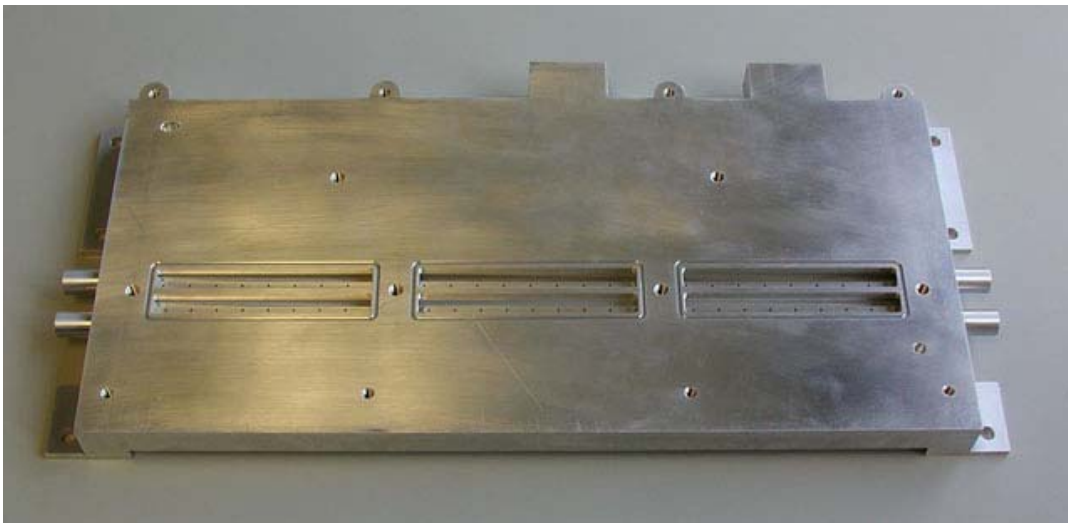


Figure 14. Base plate, with nozzle plate inserted, shown with module mounting surface up



Figure 15. Close view of the nozzles, elongated holes, and O-ring grooves

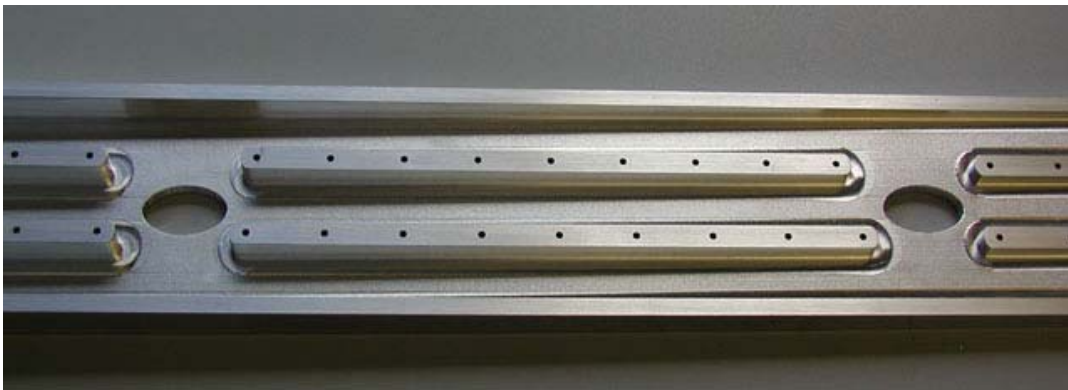


Figure 16. View of the center of the nozzle plate, showing nozzle placements, holes for mounting bosses, and the angled outlet plenum



Figure 17. View of one end of the nozzle plate

Conclusions

A series of structural, fluid, and thermal simulations performed on various geometries for the nozzle plate and base plate has resulted in a design that is ready for prototype evaluation. Experimental analysis of the major influences on thermal performance of the multijet concept showed the effects of nozzle size, distance from the target, and fluid flow rate. Simulation and experiment resulted in a design that allows use of 105°C coolant at 100 W/cm² heat flux and keep die temperatures at acceptable levels and pressure drop approximately the same as Semikron's current elliptical fin product.

Next Steps

After Semikron adds the electronics module, the fully functioning prototype will be tested in NREL in a long-term evaluation of thermal, fluid, and structural performance. Further modifications to the nozzle plate and base plate may be necessary as the test progresses. The erosive and corrosive effects of new and old coolant will be explored. Collaboration with Semikron will continue as a design ready for production is developed.

References

D. Benhammou, Jet Impingement Experiment Study, NREL Technical Report, June 2006

Patents

Patent pending.

2.6 Dynamic and Steady State Modeling to Identify, over Various Drive Cycles, Component and System Performance Efficiency and Thermal Loads

Principal Investigator: Michael P. O'Keefe

National Renewable Energy Laboratory

1617 Cole Blvd.

Golden, CO 80401 USA

Voice: 303-275-4268; Fax: 303-275-4415; E-mail: michael_okeefe@nrel.gov

DOE Technology Development Manager: Susan A. Rogers

Voice: 202-586-8997; Fax: 202-586-1600; E-mail: Susan.Rogers@ee.doe.gov

Abstract

This report summarizes the FY 2006 activities in dynamic and steady state modeling applied to Advanced Power Electronics and Electric Machines (APEEM) thermal control technology R&D. It elaborates on two key areas of focus: (1) the development of thermal control requirements by cascading vehicle level requirements and duty cycles down through APEEM components. A survey of the various powertrain technologies of interest to DOE (hybrid electric vehicle, fuel cell vehicle, and plug-in hybrid electric vehicle) has been conducted from a thermal control standpoint; and (2) the valuation of APEEM thermal control technologies in a systems context. A heat transfer model of the inverter power electronics board and assembly is used to compare the various cooling technologies for their performance.

Objective

Guide the Advanced Power Electronics and Electric Machines (APEEM) Program's thermal control technology R&D effort to best meet APEEM program needs by:

- Defining the thermal requirements for advanced automotive powertrain systems of interest to DOE and FreedomCAR: hybrid electric vehicle (HEV), plug-in hybrid electric vehicle (PHEV), and fuel cell vehicle (FCV).

- Evaluating thermal control technologies in a systems context.

- Recommending a thermal control R&D pathway.

Approach

Analyze the thermal control portfolio of R&D projects to:

- Define the Thermal Cooling Requirements for Advanced Powertrains: Use a vehicle system's context to explore the cooling requirements of various powertrain architectures.

- Evaluate Thermal Control Technologies: Work with project principal investigators and use computational analysis to compare and contrast the thermal control technologies under investigation or consideration in the DOE program. These comparisons should fall within logical

and consistent contexts (i.e., tied back to actual component operation and accepted worst-case scenarios) that show an “apples-to-apples” comparison of cooling performance, implications for component life, cost, volume, and weight.

Major Accomplishments

Preliminary identification of the thermal requirements for power electronics and electric machines (PEEMs) in advanced vehicle powertrains such as PHEVs.

Evaluation of the APEEM portfolio of thermal control technologies in a systems context with an inverter assembly thermal model.

Technical Discussion

We focused on two topic areas during FY 2006: preliminary identification of the thermal requirements of advanced powertrains, especially PHEVs; and the evaluation of the APEEM portfolio of thermal control technologies in a systems context.

PEEM System Thermal Requirements

The thermal requirements for PEEMs, especially with emerging technology and new vehicle requirements, are developing. This study yields a preliminary estimate of the additional heat load that must be dissipated for advanced power train systems (HEVs, PHEVs, and FCVs) as a first step in identifying these requirements. Special attention is paid to PHEVs. This section begins with a background on PHEVs and follows with a discussion of the analysis conducted to estimate the heat rejection and cooling requirements for PHEVs, HEVs, and FCVs.

Background on Plug-in Hybrid Electric Vehicles

PHEV is a dual-fuel technology that can transform the transportation energy infrastructure from nonrenewable, high-carbon fuels to more environmentally responsible options. One PHEV fuel is electricity from the electrical utility grid (a secondary energy source generated from one or more primary energy sources such as coal, natural gas, wind, and hydro power). The other fuel could be one of any number of options, although gasoline is most likely. This is in contrast to the conventional HEV, in which all of the motive power ultimately derives from gasoline (electricity is used only to temporarily store energy).

PHEVs can deliver performance equivalent to today's vehicles and offer key advantages related to energy security and transportation sustainability. Compared with other technology options, PHEVs do not have the infrastructure issues (e.g., hydrogen for FCVs) or the limited range issues (e.g., battery electric vehicles [BEVs]) exhibited by other technologies. The energy security and fuel efficiency benefits of PHEVs result from efficient delivery of fuel energy from the tank to the wheels and, more importantly, a transition from conventional transportation fuels to electricity. This is possible because PHEVs exhibit aspects of BEVs and HEVs, namely:

A large energy storage unit that can be recharged from the electrical utility grid and supply net motive energy over a significant distance.

A hybrid powertrain that typically uses an internal combustion engine with an electric motor.

By blending aspects of BEVs with conventional HEVs, we can realize many of the advantages of a BEV and eliminate several disadvantages. Because a PHEV requires fewer batteries than a full BEV, the PHEV comes with a reduced cost penalty versus a BEV for equivalent performance. Furthermore, the PHEV has range equivalent to an HEV or conventional vehicle (CV), and charging times are much shorter than for an equivalent BEV. In contrast to an equivalent HEV, overall fuel consumption (and liquid fuel consumption in particular) is further reduced since fuel energy is supplied from both electricity and liquid fuel as opposed to just liquid fuel, as is the case for conventional HEVs.

Although the general traits mentioned here are true for any PHEV, the specific architecture and optimal configuration of the PHEV is still in flux. Current consensus tends to favor a parallel HEV architecture with increased battery capacity. Other architectures have been proposed, however. For example, a series “range-extending” option (where the engine serves only to generate electricity for batteries) has been successfully built and sold (the Renault Kangoo).

The question of how much electrical capacity to store on-board the PHEV is also still under debate. The PHEV20 appears to be a good choice for a midsize sedan, based on some preliminary work done at NREL (Simpson et al. 2006), though PHEV40 and PHEV60 vehicles are also being considered. (The 20, 40, and 60 refer to the distance that can be driven in miles all-electrically by the PHEV.) The issue of enforcing an all-electric range on the PHEV (a range where the engine would absolutely not operate) is still open, though early analysis indicates that an all-electric range constraint may add significant cost to a PHEV at minimal additional benefit.

PHEVs work well with duty cycles where many relatively short trips comprise the bulk of distance traveled. By recharging between these short trips, a large portion of the motive energy to propel the vehicle can come from the electrical grid instead of gasoline or other fossil fuels. The transition from today’s petroleum-based transportation fuels to electricity opens up many opportunities. By recharging the vehicles’ batteries overnight, electrical utilities can increase their operating efficiency by increasing their base load. Furthermore, the difference between peak capacity and base-load capacity means that power utilities will typically have enough excess capacity at night to recharge a large number of vehicles before they have to add new generating capacity (new power plants). If done correctly, this can lead to net reductions in total carbon dioxide and other greenhouse gas emissions. Furthermore, if effort is made to transform the power plants on the national electrical grid to use more renewable sources of energy, the benefits of renewable energy can be brought to the transportation sector directly through PHEVs. All of this together will have a positive impact on national energy security, energy sustainability, and environmental stewardship.

However, PHEVs face major challenges to commercialization. Chief among these is increased vehicle cost, which results from the large number of advanced batteries on the vehicle, the life of those batteries, the hybrid-electric drive system (PEEMs), and supporting electrical subsystems. For other recent analyses on PHEVs, see Simpson et al. (2006), O’Keefe and Markel (2006), and Markel and Simpson (2006).

To address the cost issues, engineers and scientists are seeking electric drive solutions that improve in several key areas at once: lower cost, higher efficiency, lower weight, smaller size, higher reliability, and longer life. As higher power demands are forced through smaller surface areas with warmer integrated cooling fluids near temperature-sensitive electronics, thermal control of these systems becomes increasingly critical. To make real progress on systems that advance on all the key metrics, a systems solution will be needed that closely integrates thermal control from the beginning.

In this study, we analyze the electric powertrain components in a vehicle context to determine thermal control requirements for a PHEV and other advanced architectures. This is a first step toward understanding the problem so that we can choose the best solution to solve it.

From Vehicle to APEEM Thermal Requirements

For research to be beneficial, we need to understand the context of the problem that needs to be solved. This is no less true for the research being conducted on thermal systems for PEEMs. Why? Simply because the thermal control technologies we research are subsystems of a larger electric drive system—which itself is a subsystem of the entire vehicle. A thermal control system has meaning only in the context of cooling requirements for given components. Furthermore, we must look not only at our specific context, but must also be aware of how the thermal control subsystem interacts and interfaces with other subsystems (for example, the engine thermal control system).

The FreedomCAR Program has done a good job of identifying research targets for vehicle systems. Many of these research goals have been cascaded down to the subsystem level. In this study, we will use the concept of cascading to relate vehicle-level requirements and duty cycle operation to the subsystem level. This will give us a first look at the thermal control requirements for electric drive system components for whichever powertrain architecture we care to examine.

Let us consider a midsize sedan with performance competitive for the North American market. Reasonable assumptions have been identified for the midsize sedan by NREL researchers in collaboration with the DOE/FreedomCAR Vehicle Systems Technical Team. These performance requirements appear in **Table 1**. The requirements are assembled from a database of present-day vehicle performance requirements and were identified as reasonable by the DOE/FreedomCAR Vehicle Systems Analysis Technical Team.

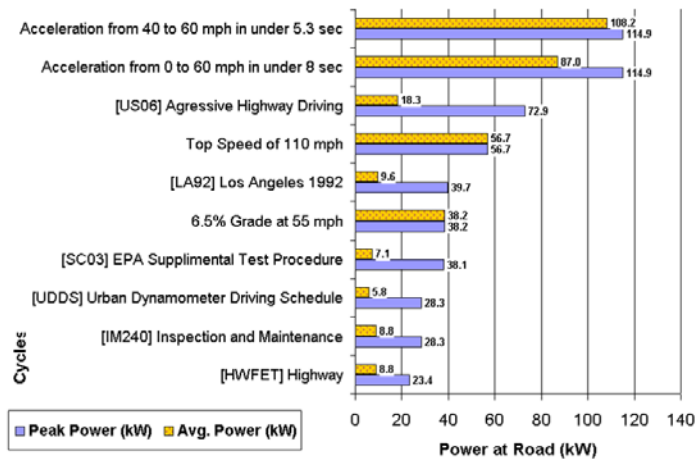
(a) **Table 1: Midsize Sedan Performance Requirements and Vehicle Assumptions**

1 Attribute	2 Value
Top Speed	110 mph
Full Acceleration time from 0 to 60 mph	≤ 8 s
Passing Acceleration time from 40 to 60 mph	≤ 5.3 s
Hill Climbing road grade (percent rise over roadway run) to climb with engine at 66% of rated power	6.5% at 55 mph
Range maximum distance traveled starting fully fueled	400 miles
Glider Mass the mass of the vehicle minus the powertrain	905 kg (1995 lb)
Accessory Loads accessory loads assumed for the PHEV	0.7 kW electric (average) 4.0 kW electric (peak)
Transmission Efficiency efficiency of the mechanical gearing between the motor/engine and wheels	85%
Electrical Generation Efficiency for Accessories efficiency of generating power to the electric accessories	85%

Figure 1 shows the average and peak power requirements at the road-tire interface to propel a vehicle with the same weight and chassis/body parameters (drag, frontal area, etc.) as the Toyota Prius over several driving events. The Prius was chosen for this study as it has become something of a baseline for HEVs. In addition to the vehicle performance requirements (acceleration, hill climbing, etc.), the power requirements for several standard vehicle duty cycles (vehicle time-speed driving patterns) are also shown. The cycles include the HWFET (highway federal emissions test—nonaggressive highway

driving), US06 (United States Highway 6—aggressive highway driving), UDDS (urban dynamometer driving schedule—nonaggressive urban driving), the IM240 (inspection and maintenance for 240 seconds—a light urban driving cycle), and the LA92 (Los Angeles 1992—aggressive urban driving). Minimum powertrain component sizes can be estimated based on these duty cycles and performance requirements.

Because even a PHEV cannot power a vehicle continuously with electricity over the vehicle's maximum driving range, there is necessarily a distinction between the vehicle's minimum engine power (which can run continuously over the vehicle's range) and minimum motor power (where continuous power is tied to the battery capacity). If we examine the duty cycle and performance constraints in **Figure 1** and apply our assumptions about transmission efficiencies and auxiliary loads from **Table 1**, we can estimate the peak and continuous power required at the engine/motor. This appears in **Table 2**.



(b) Figure 1. Vehicle minimum peak and continuous power requirements

(c) Table 2. Peak and Continuous Power Requirements

$P_{peak,min} = \frac{114.9 kW}{0.85} + \frac{4 kW}{0.85} \approx 140 kW$ based on acceleration requirement; assumes that top speed is a peak power requirement
$P_{continuous,min} = \frac{\frac{38.2 kW}{0.85} + \frac{4 kW}{0.85}}{0.66} \approx 75 kW$ based on hill climbing requirement where power can only be 66% of engine peak
$P_{peak} = P_{engine} + P_{motor,peak} \geq 140 kW$ $P_{continuous} = P_{engine} \geq 75 kW$ $\therefore$ if $P_{motor,peak}$ is 55 kW as per FreedomCAR targets, P_{engine} must be $\geq 85 kW$

The performance requirements listed in **Table 1** imply certain minimum continuous and peak power outputs for vehicle powertrain components, which are summarized in **Figure 1**.

According to the conservative analysis given in **Table 2**, an engine of 85 kW or greater power coupled with a 55 kW (peak) motor would be ideal. These power requirements are derived by applying the performance requirements from **Table 1** to a vehicle with the same weight and body parameters (drag coefficient and frontal area) as the Toyota Prius. In actuality, the peak power output from the Toyota Prius powertrain is 82 kW at 53 mph—substantially less than the 140 kW peak predicted here. This means that the actual Toyota Prius does not meet the performance requirements of **Table 2**. A quick search on the Internet for acceleration times of the Prius will confirm this is true. This raises the question of which high-level design requirements we should work toward. Aggressive performance requirements add more cost and system complexity. However, some minimum performance will be needed for consumer acceptance. **Table 1** is indicative of the North American light-duty vehicle market—but is it too aggressive? The Prius, despite meeting less stringent market requirements, has some degree of consumer acceptance.

The previous analysis shows how vehicle requirements and duty cycles relate to component sizes. We have confirmed that the FreedomCAR motor peak power shaft output of 55 kW is reasonable and that an engine size of 85 kW shaft output is in the ballpark for a car the size of a Toyota Prius with North American market performance (assuming a parallel hybrid configuration). Let us shift now to a study of how vehicle duty cycle and performance requirements relate to electric drive component operation.

The ADVISOR™ software results from the simulation of several midsize sedan vehicle models are presented next. The objective of this preliminary study is to have a sense for the heat rejection profiles of components in electric drive subsystems of advanced powertrains over the execution of various vehicle duty cycles and performance requirements. The numerical vehicle models used in this analysis are representative of midsize sedan HEV, PHEV, FCV, and CV architectures. The models were created by NREL's Vehicle Systems Analysis Team for a separate analysis and were useful for our analysis as well. We intend to continue—and expand—the collaboration with the DOE FreedomCAR Vehicle Systems Analysis Team to ensure we are using the best, most accurate, and most relevant vehicle models and data possible.

Each numerical vehicle model in this study uses a different powertrain: CV (manual transmission), HEV (parallel), PHEV (similar to the HEV but with larger size and capacity components), and FCV. By

simulating each vehicle model over several duty cycles (speed-time traces), we can learn how the electric drive duty cycle is influenced by the vehicle-level duty cycle.

An overview of the vehicle models used in this study is given in **Table 3**. Although the performance of each model is similar to that presented in **Table 1**, the FCV model in particular has not been rigorously adjusted for performance parity. Thus, the FCV model has less aggressive performance than that specified by the requirements. NREL intends to continue working with other groups in the DOE program on vehicle systems modeling to improve these models. For now, we present the models we have from the ADVISOR software, which should give some early directional indications for component heat rejection.

(d) **Table 3. Parameters for Vehicle Models Used in This Study**

3 Attribute	4 CV	5 HEV	6 PHEV20	7 PHEV40	8 FCV
Weight (lb/kg)	3587/ 1627	3384/ 1535	3525/ 1599	3616/ 1640	3706/ 1681
Drag Coefficient	0.30	0.30	0.30	0.30	0.33
Frontal Area (m ²)	2.27	2.27	2.27	2.27	2.00
Accessory Loads (kW) (mechanical/electrical)	0.823/ 0.0	0.0/ 0.7	0.0/ 0.7	0.0/ 0.7	0.0/ 0.7
Rolling Resistance Coefficient	0.009	0.009	0.009	0.009	0.009
Prime Mover Peak Power (kW)	145	82	84	86	58
Motor Power (kW) (Peak/Continuous)	NA	39/ 39	50/ 50	50/ 50	88/ 88
Battery Capacity [usable] (kWh)	NA	0.38	5.67	10.84	0.73
Top Speed (mph)	147.5	137.1	134.4	138.9	97.9
Acceleration Time 0–60 mph (seconds)	8.0	8.1	8.0	8.1	10.9
Fuel Economy over UDDS Cycle (mpg gasoline equivalent) (charge sustaining mode)	22.7	37.4	38.3	37.6	70.1

Each powertrain considered has vastly different thermal performance (see **Figure 2**). Here the average heat rejection by subsystem is listed for the various powertrains over the UDDS cycle. The CV, HEV, PHEV20 (a PHEV with 20 miles of all-electric driving capacity), PHEV40 (a PHEV with 40 miles of all-electric driving capacity), and FCV are shown. The CV, because of its large internal combustion engine, rejects much more heat than the other vehicles. A large portion of this heat is rejected directly to the atmosphere through engine exhaust. Most of the remaining energy is rejected via the engine coolant through the radiator. Thus, despite the large amount of heat rejected by the internal combustion engine in a CV, the engine, which can withstand relatively high temperatures (compared to power electronics) has no major thermal control issues (i.e., the technology adequately cools present designs).

In contrast, the power electronics for HEVs and PHEVs reject much less heat on average than the internal combustion engine; however, it must reject that heat through much smaller thermal masses, within confined spaces, and in the presence of temperature-sensitive electronics. This situation presents a major technical hurdle to technology advancement in PEEM R&D. It also emphasizes the need for focused thermal cooling R&D as a joint effort with the PEEM R&D.

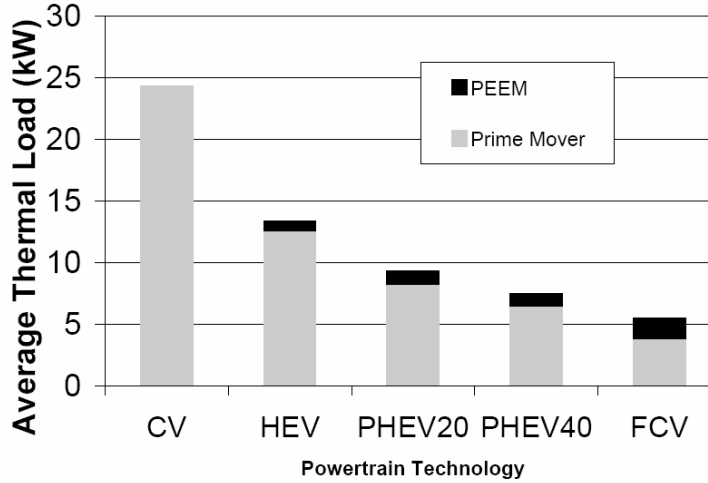
Because PHEVs are of special interest, we will now look more specifically into the cooling requirements for PHEVs in contrast with HEVs. **Table 3** shows that the motor size for the PHEV is 25% larger than that of the HEV. A first approximation of the increase in thermal loads of a PHEV over an HEV would be:

(e) **Equation 1:**
$$PITL = \frac{\overline{ML}_{PHEV}}{\overline{ML}_{HEV}} \cdot \frac{(P_{\max, motor, PHEV} - P_{\max, motor, HEV})}{P_{\max, motor, HEV}} \cdot 100\%$$

where:

$PITL$ = percentage increase in thermal loads and

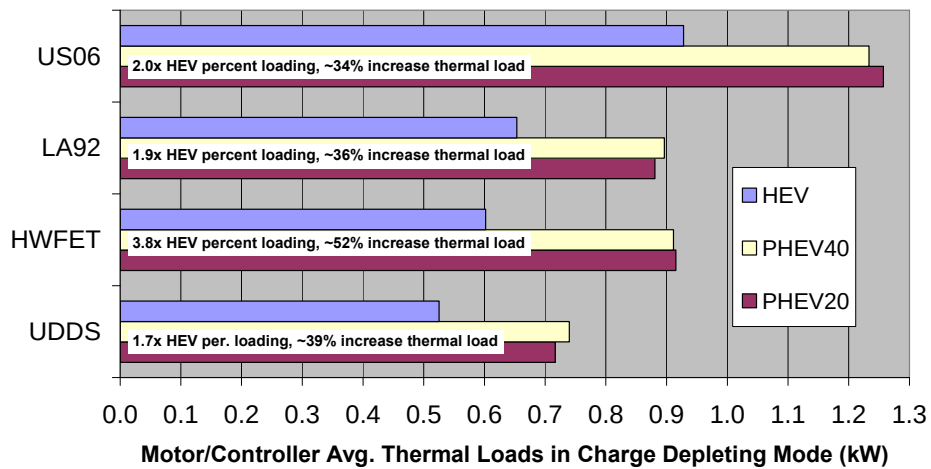
ML = motor loading =
$$\frac{\overline{P}_{motor}}{P_{\max, motor}}$$



(f) **Figure 2. Average thermal loads by vehicle subsystem and vehicle powertrain (UDDS cycle)**

However, when we apply **Equation 1**, we find that the percentage increase in thermal loading of PEEM systems for the PHEV versus the HEV is overestimated (see **Figure 3**). As noted on the graph, for the US06 aggressive highway driving cycle, the average motor loading of the PHEV is twice that of the HEV. That is, for the PHEV, the average to peak possible power throughput at the motor is twice that of an HEV. Coupled with a 25% increase in motor size for the PHEV versus HEV (given in **Table 3**), we would have expected a 50% increase in thermal loads; however, after running our vehicle system models, we see that there has been only a 36% increase in thermal loads. Why is this?

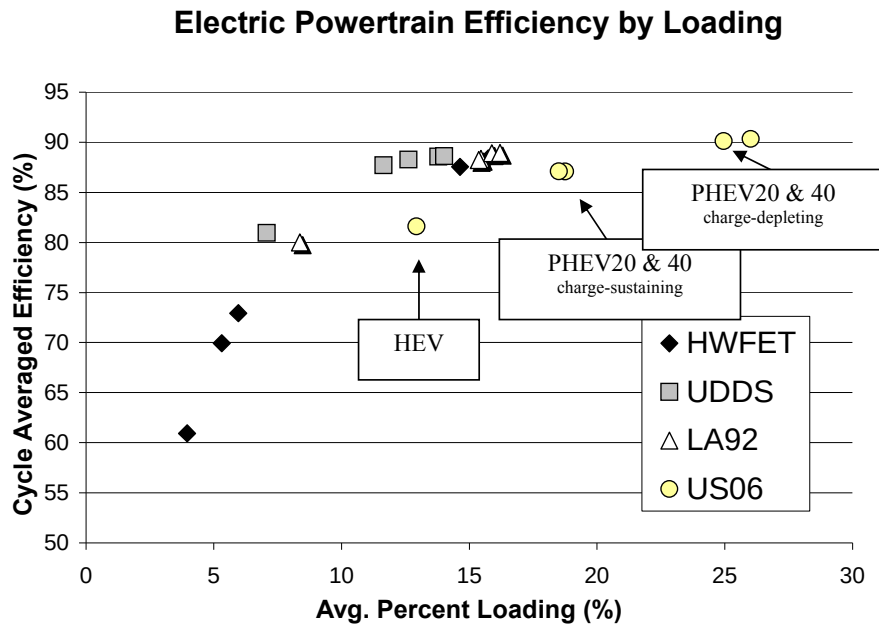
The simple relationship given by **Equation 1** for the increase in thermal loading is linear. However, we are dealing with a nonlinear system. Specifically, we see from **Table 4** that as the PEEM drive system utilization is increased (i.e., as the system reaches higher average loading), the system efficiency increases. This increase in efficiency counteracts the increase in wasted heat that would have been seen otherwise. This is clearly demonstrated in **Figure 4**.



(g) Figure 3. Increase in PEEM thermal loads for PHEVs over HEVs

(h) Table 4. Motor and Inverter Combined Heat Rejection by Duty Cycle

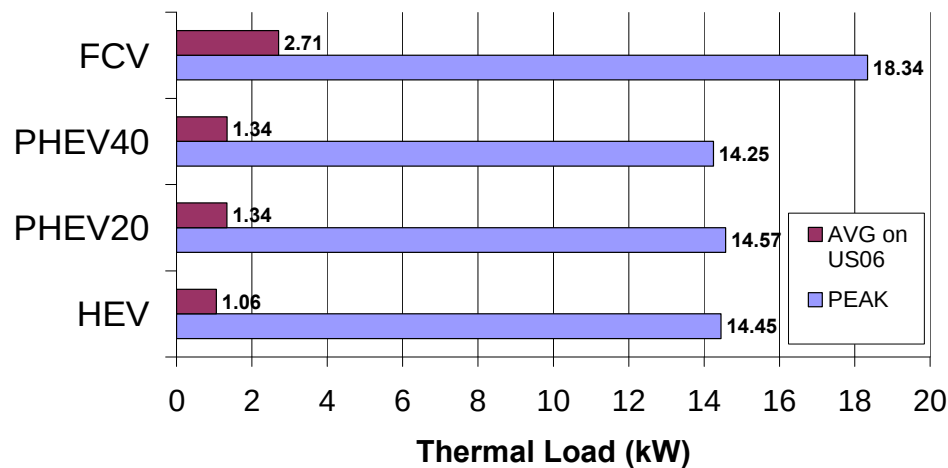
9 Powertrain	10 $\overline{P}_{loss}$	11 $\overline{P}_{shaft,out}$	12 $\overline{P}_{electric,in}$	13 $\frac{\overline{E}_{shaft}}{D}$	14 $\overline{\eta}_{shaft,out}$
15 –	16 kW	17 kW	18 kW	19 kJ/mile	20 %
UDDS Cycle (1369 s and 7.45 miles per cycle)					
HEV	0.53	2.23	2.76	115.5	81.0
PHEV20 charge-sustaining	0.79	6.11	6.90	420.6	88.6
PHEV20 charge-depleting	0.72	5.11	5.82	595.1	87.7
PHEV40 charge-sustaining	0.80	6.21	7.01	428.6	88.6
PHEV40 charge-depleting	0.74	5.57	6.31	615.9	88.3
HWFET Cycle (765 s and 10.26 miles per cycle)					
HEV	0.60	0.94	1.54	5.8	60.9
PHEV20 charge-sustaining	0.81	2.17	2.98	46.5	72.9
PHEV20 charge-depleting	0.92	6.96	7.87	462.4	88.4
PHEV40 charge-sustaining	0.80	1.86	2.65	35.1	69.9
PHEV40 charge-depleting	0.91	6.41	7.32	415.1	87.5



(i) Figure 4. PEEM cycle averaged efficiency by loading

Figure 4 shows the average percent loading of the HEV, PHEV20, and PHEV40 models simulated over various drive cycles. The average percent loading is the average power output at the motor shaft divided by the maximum possible shaft output power times 100%. The PHEVs are simulated in charge-sustaining mode (where the net energy in the energy storage system is relatively constant over distance) and charge-depleting mode (where there is a net reduction in energy storage system energy with distance). Figure 4 demonstrates a general trend for the increase in cycle averaged efficiency as we move from an HEV to a PHEV. The strength of this trend varies over duty cycle, but is visibly present over all the driving profiles examined. This favorable increase in cycle averaged efficiency decreases the average heat rejection from the PEEM systems for PHEVs compared to HEVs over the various drive cycles. This trend does not hold consistently for PHEV charge-sustaining versus charge-depleting operation. In fact, a PHEV charge-depleting operation is not always more efficient than a PHEV charge-sustaining operation. This depends on the specifics of the given duty cycle (see **Table 4**).

The analyses presented thus far examine cycle averaged thermal loading, which tells us about the anticipated day-to-day average heat rejection of the PEEM components. This can relate to life and daily performance. However, peak—not average thermal loading—must be used to size PEEM thermal control technologies. Peak losses are compared to average losses in **Figure 5**. The peak losses are the maximum thermal load generated over either the US06 or 0–60 mph acceleration test. The average losses correspond to the US06 (the average thermal load per time over the US06). There is a huge disparity between the peak and average loads. This disparity indicates that technologies such as thermal storage technologies (phase change materials and the like) that remove the peak rejection rates and “level the load” may be worthwhile to examine for use in a thermal control system. A thermal storage technology could enable cheaper and smaller thermal control devices sized for average versus peak power rejection.



(j) Figure 5. Average versus peak thermal loading from PEEM components

Analysis of Thermal Control Technologies

The second major effort conducted during FY 2006 was to evaluate some of the current APEEM Thermal Control R&D in a systems context. Specifically, we have estimated the peak silicon device temperature (the IGBT or diodes) in inverter power electronics that are associated with each thermal control technology under various heat rejection scenarios. A one-dimensional (1D) heat-transfer model and a more sophisticated three-dimensional (3D) finite element model (FEA model) of the inverter power electronics are used to simulate the peak silicon device temperatures and to make consistent comparisons of the APEEM thermal control technologies. This next section will present the heat transfer models as well as the results thus far on this effort.

Inverter Power Electronics Heat Transfer Models

Two heat transfer models were used in this study: a 1D heat transfer model and an FEA model. The fundamentals of both models can be understood by considering the 1D electrical-circuit analogy to heat transfer. In the electrical-circuit analogy, the concept of thermal resistance is modeled analogous to “electrical resistors.” Temperature is analogous to voltage, and temperature drops analogous to voltage drops. Entropy flow (heat transferred at a given temperature) becomes analogous to current.

Resistance to thermal conduction is given by the relation:

(k) **Equation 2:** $\theta_{th,conduction} = \frac{L}{k \cdot A}$

(l) **Equation 3:** $R_{th,conduction} = T \cdot \theta_{th,conduction}$

where θ is “classical thermal resistance,” R is an “Ohm’s law thermal resistance,” L is the thickness of the layer, k is the thermal conductivity (in units of W/(m·K) or similar), T is the temperature of the material, and A is the cross-sectional area for convective heat transfer. In SI, the units are K²/W for the Ohm’s law thermal resistance and K/W for classical thermal resistance. There is a similar relationship for the resistance associated with convective heat transfer:

(m) **Equation 4:** $\theta_{th,convective} = \frac{1}{h \cdot A}$

(n) **Equation 5:** $R_{th,convective} = T \cdot \theta_{th,convective}$

where h is the convective heat transfer coefficient in W/(m²·K) and A is the corresponding area for heat transfer. Again, T shows up as the bulk temperature associated with the resistance and thermal resistance can be given by either the “classical” form or the Ohm’s law form.

Entropy flow is analogous to current:

(o) **Equation 6:** $\dot{S} = \frac{\dot{Q}}{T}$

where $\dot{Q}$ is the 1D rate of heat transfer through a material (in W) and T is the bulk representative temperature in Kelvin of the material. $\dot{S}$ is the entropy flow.

To determine temperature drops across a given thermal layer, we can use the analogy of Ohm’s law as follows:

(p) **Equation 7:** $\Delta V = I \cdot R \rightarrow \Delta T = \dot{S} \cdot R_{th}$

which can be simplified to:

(q) **Equation 8:** $\Delta T = \dot{Q} \cdot \theta_{th}$

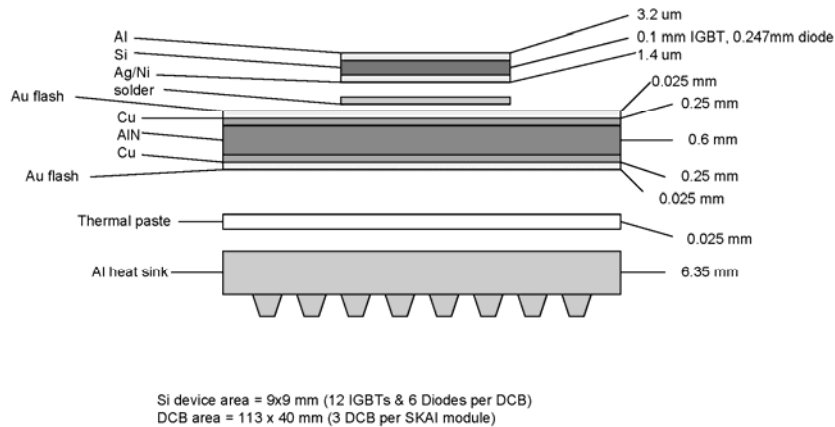
where θ_{th} would correspond to either the convective or conductive version of classical thermal resistance, depending upon the context of where one was measuring the temperature drop.

Similar electrical relationships apply using the variables we’ve defined. For example, the power attributed to heat rejection through any given layer is:

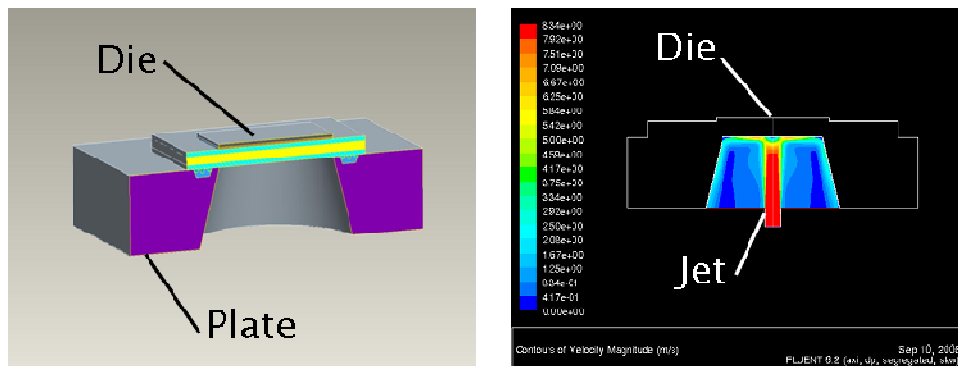
(r) **Equation 9:** $P = I^2 \cdot R = I \cdot V \rightarrow \dot{Q} = \dot{S}^2 \cdot R_{th} = \dot{S} \cdot T$

The electrical analogy to heat transfer serves as the basis to construct a model of the heat transfer in an inverter. The inverter assembly’s various layers and components are modeled as thermal resistors that conduct heat from the silicon devices of the power electronics to the coolant of the thermal control system. The coolant in the thermal control system ultimately convects heat away to the radiator and then to the ambient air. The baseline inverter assembly is modeled after the Semikron SKAI system (see **Figure 6**). The only difference between the models we used and that of **Figure 6** is that the diode thickness is set

the same as the IGBT thickness. Thus, all silicon devices (silicon die) rise to the same peak temperature in our models. Another promising assembly structure is the low thermal resistance assembly, which opens holes in the cold plate to allow cooling via liquid coolant jets that impinge directly on the backside of the direct bonded copper (DBC) layer of the inverter. This assembly dramatically reduces the thermal conduction resistance between heat source and sink. A schematic of this layout is given in **Figure 7**.



(s) **Figure 6.** Semikron SKAI thermal stack-up (graphic courtesy of Semikron/John Mookken)



(t) **Figure 7.** Low thermal resistance assembly that allows for direct backside cooling via jets

The physical layers of the inverter form thermal resistances that result in temperature increases to conduct a given amount of heat. Another critical resistance that must be accounted for is “spreading resistance,” which is encountered when one cross-sectional area of heat flow interfaces with a different cross-sectional area of heat flow. The heat flow path is thus no longer directly vertical; instead, the heat must spread out from the smaller device to the larger. The temperature rises under these conditions. Several authors (Cullham et al. 2000), (Muzychka, 2004), and (Yovanovich et al. 1999) have introduced analytical methods to simulate spreading resistance with 1D models. The techniques presented by these authors are being investigated for use in our 1D model.

Spreading resistance is defined as follows:

(u) **Equation 10:** $\theta_{th,total} = \theta_{th,1D} + \theta_{th,spreading}$

where $\theta_{th,total}$ is the total thermal resistance, which is composed of the 1D thermal resistance ($\theta_{th,1D}$) and the spreading resistance ($\theta_{th,spreading}$).

The spreading resistance is quite large (on the same order as the total 1D resistance) and is thus important to include in the model. However, spreading resistance is a complex function of the geometry and heat transfer boundary conditions of the problem. We currently use a 3D FEA model, which is created with the ANSYS FEA software, to estimate the spreading resistance. We are still interested in the 1D model as well, though methods for including spreading resistance in the 1D model must be developed before we can use it with confidence. The 1D heat transfer model is important for its simplicity and speed. By keeping the model at a high level, we retain the ability to do quick-feedback simulations. We intend to enhance the 1D model by including spreading resistance and transient behavior (i.e., thermal “inductance” and “capacitance”) in the coming fiscal year. The objective will be to accurately predict the peak temperature increase of power electronics devices under various conditions and estimate the temperature loading on power electronics over a duty cycle.

For this report, temperatures are being predicted accurately with a model developed with the ANSYS FEA software. This model includes spreading resistance. A comparison of simulations of a baseline pin-fin heat exchanger with a 100 W/cm² thermal load on all silicon devices (IGBTs and diodes) for the architecture are shown in Figure 6 and in Table 5. This analysis compares the current predictive capability of the 1D model to the equivalent ANSYS 3D FEA model with and without spreading resistance. The 1D thermal model agrees well with the more advanced ANSYS FEA model when thermal spreading is neglected. In fact, most of the differences between the models are likely due to slight differences in assumptions between the two models. However, when spreading resistance is included, the simulated peak temperature changes significantly from the 1D case, indicating the importance of including the spreading resistance.

(v)

(w) **Table 5. 1D Model Results versus ANSYS Model with and without Thermal Spreading Resistance**

21	22 1D Thermal Model	23 ANSYS Model without Spreading	24 ANSYS Model with Spreading
$T_{coolant}$ (°C)	70.00	70.00	70.00
$T_{peak, IGBT \& Diodes}$ (°C)	97.30	97.07	114.29
ΔT (°C)	27.30	27.07	44.29
$\dot{Q}''$ (W/cm ²)	100	100	100
$\dot{Q}$ (kW)	1.458	1.458	1.458
$\theta_{th, convection}$ (°C/kW)	4.42	4.42	2.96
$\theta_{th, 1D}$ (°C/kW)	14.30	14.15	14.15
$\theta_{th, spreading}$ (°C/kW)	0.00	0.00	13.27
$\theta_{th, Total}$ (°C/kW)	18.72	18.57	30.38

Thermal Cooling Technology Assessment

In this section, we present the results of several thermal control technologies evaluated in the context of a heat transfer model of inverter power electronics. As mentioned at the end of the previous section, the

ANSYS FEA software is being used to accurately include spreading resistance. A list of the technologies investigated appears in **Table 6**.

The total classical thermal resistance from the silicon die to the coolant (bulk temperature) in our inverter assembly follows:

(x) **Equation 11:** $\theta_{total} = \theta_{th,convective} + \theta_{th,conduct,1D} + \theta_{th,spreading}$

The terms of **Equation 11** are reflected in the columns of **Table 6** and provide some insights into what each cooling technology is doing and where each can be improved. As mentioned earlier, the spreading resistance is nearly equivalent to the 1D resistance for all the technologies. Thus, any of the technologies can benefit from advanced heat spreading technology.

Looking specifically at pin fin technology, we see that, although the resistance at the pin fins is the smallest of the three terms (convective, 1D conductive, and spreading), it is still on the same order of magnitude as the other resistances. Thus, the pin fin technology can benefit from a more advanced mechanism to convect heat away at lower resistance. The jet spray work is an answer to this need to decrease pin-fin convective resistance. Advances to the thermal interface materials (TIMs) used on pin-fin devices have a significant effect on the 1D conduction resistance because the 1D thermal resistance is quite large.

When we compare pin-fin technology to liquid jets, we can see an order of magnitude decrease in the convective resistance. The 1D conductive resistance is also significantly reduced, though this is due to the low thermal resistance assembly (see **Figure 7**). Because little TIM material is used in the low thermal resistance assembly and because of the nature of the geometry, improvements to TIMs do not significantly affect the temperature increase. This can be seen by examining the change in 1D conduction resistance and thermal spreading resistance for the liquid jets when advanced TIMs are applied. The 1D conduction resistance does drop. However, at the same time, the spreading resistance increases. A multipronged approach that uses advanced TIMs to decrease 1D resistance and simultaneously attacks spreading resistance may prove more effective.

Looking at the possibilities with air cooling, we see first of all, that under very optimistic assumptions for future performance, air cooling has nearly doubled the total thermal resistance of the pin fin technology. However, air cooling's saving grace is the coolant temperature—assumed here to be 30°C. Thus, even with the higher resistance, maximum temperature increases are acceptable for air under the optimistic scenario.

We have a long way to go before achieving the highly optimistic air cooling performance given under the “best possible” scenario of **Table 6**. The present day cooling capability of air is listed as “Air Jets (present day)” in the table. The maximum temperature increase is clearly unacceptable. The major challenge with air will be to decrease the cooling resistance to acceptable levels. If levels similar to that given by the “best possible” case can be achieved, other improvements to 1D conductive and spreading resistance may be beneficial.

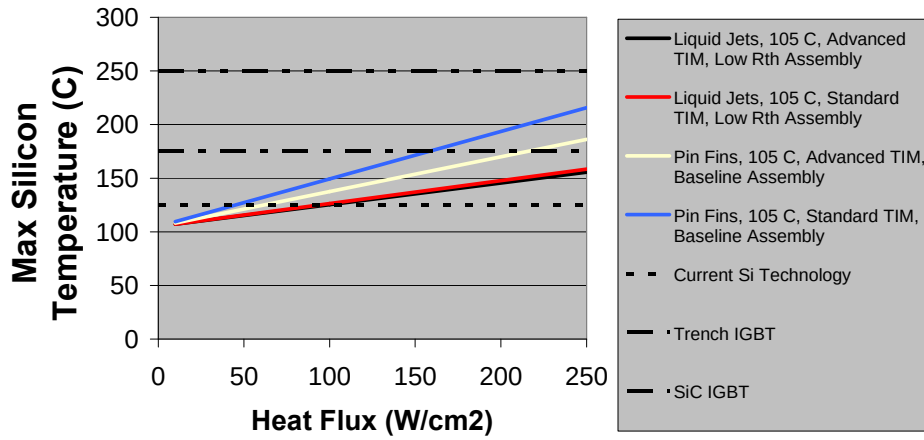
Figure 8 through **Figure 10** show the results as a function of heat rejection rate per unit area ($\dot{Q}''$). These results are given for three coolant temperature assumptions: 30°C, 70°C, and 105°C. These assumptions come from estimated coolant temperatures for air cooling, a single loop (cooling only the PEEM components), and dual loop (a coolant loop for both the engine and PEEM components), respectively. The figures show liquid jets with a low thermal resistance assembly to be the most promising near-term technology. Also, the application of advanced TIMs on pin fin technology makes a significant difference. If the optimistic potential for air cooling can be approached, this technology could be quite promising as well, but there is a long way to go.

The results generated in **Table 6**, **Figure 8**, **Figure 9**, and **Figure 10** are based on numerical models created with the ANSYS FEA software. We will improve these models and do further experimental work to validate the results presented here. Also, the current simulations show only peak temperature increase. Other critical metrics such as weight, volume, cost, and the impact on component life will be addressed as this project progresses.

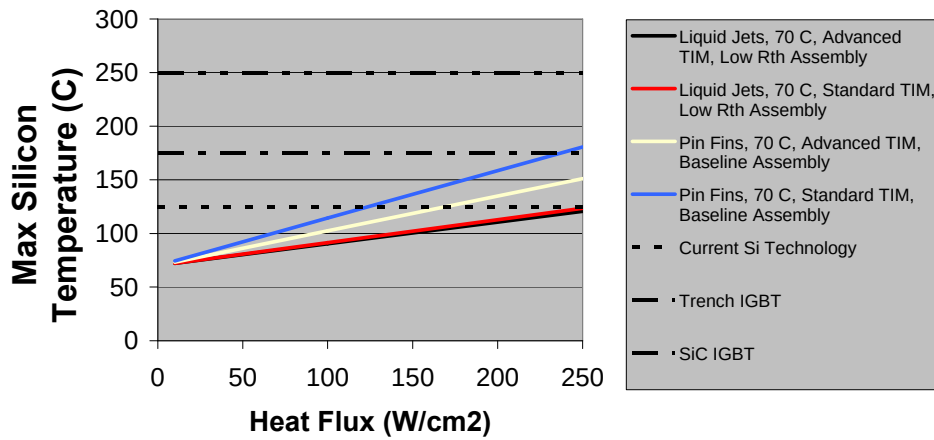
(y) Table 6. APEEM Thermal Control Technologies Evaluated

Cooling Apparatus	TIM Conduction (W/m.K)	Convection Coefficient (W/cm ² .K)	Coolant	Coolant Bulk Temperature (C)	Assembly	Silicon Device Max Temperature (C) **	$\theta_{contact}$ C/kW	$\theta_{cond,LD}$ C/kW	θ_{sp} C/kW	θ_{total} C/kW
Pin Fins	1	2	WEG	105	Baseline	149.29	2.96	14.14	13.27	30.38
Pin Fins	1	2	WEG	70	Baseline	114.29	2.96	14.15	13.27	30.38
Pin Fins	7.5	2	WEG	105	Baseline	137.43	2.79	9.35	10.10	22.24
Pin Fins	7.5	2	WEG	70	Baseline	102.43	2.79	9.35	10.10	22.24
Liquid Jets	1	6, 3	WEG	105	Low Rth	126.37	0.45	8.36	5.84	14.65
Liquid Jets	1	6, 3	WEG	70	Low Rth	91.37	0.45	8.36	5.84	14.65
Liquid Jets	7.5	6, 3	WEG	105	Low Rth	125.23	0.58	6.38	6.92	13.88
Liquid Jets	7.5	6, 3	WEG	70	Low Rth	90.23	0.58	6.38	6.92	13.88
Air Jets (best possible)	1	0.50, 0.25	Air	30	Low Rth	114.60	22.45	13.38	22.20	58.02
Air Jets (best possible)	7.5	0.50, 0.25	Air	30	Low Rth	105.69	23.29	8.49	20.14	51.91
Air Jets (present day)	1	0.04, 0.02	Air	30	Low Rth	632.35	367.68	14.24	31.22	413.13
Air Jets (present day)	7.5	0.04, 0.02	Air	30	Low Rth	620.49	368.92	8.78	27.30	405.00

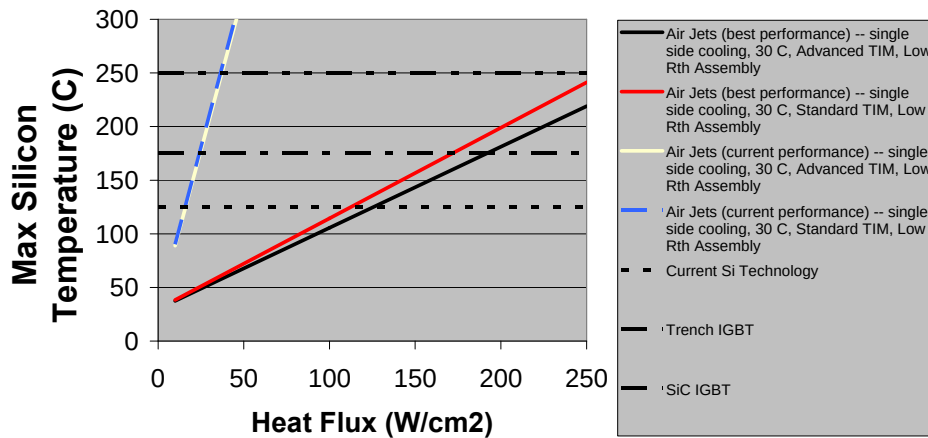
** Maximum temperature for a 100 W/cm² thermal load (1.458 W of thermal energy input)



(z) Figure 8. Si die max temperature for 105°C coolant APEEM cooling technologies by $\dot{Q}''$



(aa) Figure 9. Si die max temperature for 70°C coolant APEEM cooling technologies by $\dot{Q}''$



(bb)Figure 10. Si die max temperature for 30°C coolant APEEM cooling technologies by $\dot{Q}''$

Conclusions

In conclusion, this report has surveyed the initial work to bring a systems analysis perspective to the thermal cooling R&D effort associated with the DOE APEEM program. Two key areas were discussed:

Determination of thermal cooling requirements and thermal loads by the cascading of vehicle level requirements and duty cycle operations down through the various APEEM drive technologies

Evaluation and application of the thermal control technology portfolio in a systems context using an inverter heat transfer model to show the effectiveness of the various cooling strategies we are evaluating under the APEEM program.

Next Steps

Several next steps have been identified for this thermal subsystem analysis effort:

1. Collaborate with the FreedomCAR vehicle systems analysis team to simulate electric drive subsystems with DOE FreedomCAR-accepted vehicle models for CVs, HEVs, PHEVs, and FCVs. This work should extend into predictions for actual thermal loadings over time from inverters and other APEEM components—ongoing through FY 2007.
2. Improve the 1D thermal model of the inverter power electronics assembly to include thermal spreading. Validate the model with more advanced simulations (e.g., ANSYS) and laboratory testing—December 2006.
3. Include the ability to handle transients in the 1D thermal model of inverter power electronics. Apply this model to thermal cycling analysis—April 2007.
4. Evaluate remaining APEEM thermal control technologies with the inverter thermal model—December 2006.

5. Increase collaboration between the NREL team and other APEEM teams (specifically ORNL) to better couple the thermal design and analysis with current APEEM projects—ongoing through FY 2007.

References

Culham, J., Yovanovich, M., Lemczyk, T. “Thermal Characterization of Electronic Packages Using a Three-Dimensional Fourier Series Solution.” *Journal of Electronic Packaging*, Vol. 122. September 2000.

Markel, T., Simpson, A. “Plug-in Hybrid Electric Vehicle Energy Storage System Design.” Presented at the Advanced Automotive Battery Conference in Baltimore, Maryland. May 19, 2006.

Muzychka, Y. “Influence Coefficient Method for Calculating Discrete Heat Source Temperature on Finite Convectively Cooled Substrates.” 2004 Inter Society Conference on Thermal Phenomena. 2004.

O’Keefe, M., Markel, T., “Dynamic Programming Applied to Investigate Energy Management Strategies for a Plug-in HEV.” To be presented at EVS-22, Friday October 27, 2006.

Simpson, A., Markel, T., Brooker, A. “Cost-Benefit Analysis of Plug-in Hybrid Electric Vehicle Technology.” To be presented at EVS-22, Friday October 27, 2006.

Yovanovich, M., Muzychka, Y., Culham, J. “Spreading Resistance of Isoflux Rectangles and Strips on Compound Flux Channels.” *Journal of Thermophysics and Heat Transfer*, Vol. 13, No. 4,. October–December 1999.

2.7 Advanced Thermal Interface Materials to Reduce Thermal Resistance

Principal Investigator: Sreekant Narumanchi
National Renewable Energy Laboratory

MS 1633
1617 Cole Blvd
Golden, CO 80401-3393
Voice: 303-275-4062; Fax: 303-275-4415; E-mail: sreekant_narumanchi@nrel.gov

DOE Technology Development Manager: Susan A. Rogers
Voice: 202-586-8997; Fax: 202-586-1600; E-mail: Susan.Rogers@ee.doe.gov

Abstract

In a typical IGBT package in an inverter, the grease layer which attaches the DBC layer to the heat sink has the highest thermal resistance in the package. This report describes our attempt to develop high performance, reliable, and low cost thermal interface materials (TIMs). Three different techniques for growing nanotubes on a copper substrate have been explored. Preliminary thermal resistance measurements have been made. Metal-coated particles embedded in polymer compounds are also being explored for possible use as TIMs. A test stand is being acquired in-house to characterize the thermal resistance of these novel TIMs.

Objectives

Thermal interface materials pose a major bottleneck to heat removal from the IGBT package. In a typical IGBT package, for example, the direct bond copper (DBC) layer is attached to the aluminum heat sink via a thermal grease, which has a thermal conductivity of 0.8 W/mK and is a major contributor to the thermal resistance in the package. Accordingly, a project was undertaken in FY 2006 to explore advanced thermal interface materials for reducing the thermal resistance. These are the objectives of this project:

- Identify and assist in the development of low thermal resistance, low cost, thermal interface materials. We are currently exploring interface materials based on carbon nanotubes (CNTs) and metal coated particles.
- Characterize the thermal performance of these novel interface materials over an extended temperature and pressure range characteristic of automotive applications.
- Characterize the impact of thermal cycling on thermal resistance.

If these advanced materials are suitable for automotive applications and yield enhancements in thermal conductance over the state-of-the-art conventional interface materials by at least a factor of 5, the ultimate objective is to explore the possibility of replacing the grease in IGBT packages with these advanced thermal interface materials.

Approach

NREL's approach for FY 2006 in order to meet the objectives mentioned above is given below.

- We conducted a survey of the existing state-of-the-art thermal interface materials. This includes greases, pastes, filler pads, adhesives, polymer compounds, solders, phase change materials, as well as the state-of-the-art carbon nanotubes research.
- A subcontract was established with the University of Colorado at Boulder to explore carbon nanotubes as thermal interface materials. The idea is to grow CNTs directly on a copper substrate/foil, which could then be used as an interface material. In the literature, some studies have reported enhanced thermal conductance of the grease by up to 100% when nanotubes are embedded in the grease. The concept behind growing nanotubes directly on a substrate is to obtain even higher thermal conductances than can be obtained by enhancing thermal greases.
- After the growth process of nanotubes on the copper foil is optimized, characterize the thermal resistance of the copper foil-CNT-copper foil structure and compare with a copper foil-conventional TIM-copper foil structure.
- By the end of FY 2006, a subcontract with Federal Technology Group (FTG) will be established. FTG is a company that specializes in the metal coated particle technology for various applications. The idea is to embed metal coated particles (example—copper-coated diamond) in a polymer compound in order to enhance the thermal conductance. FTG will also be working with companies such as Honeywell and GE. The idea behind metal coated particles is to reduce the interfacial thermal resistance between the particles and the host matrix. These metal coated particles also have very good wetting properties. Again, the desire is to obtain at least a factor of 5 enhancements over the state-of-the-art thermal interface materials.
- We are close to acquiring in-house at NREL a test stand for performing thermal resistance measurements over the automotive temperature range (-40°C – 150°C). This test stand is based on the standard ASTM-D5470 test method for thermal resistance measurements. This test stand will be utilized to perform thermal resistance measurements of the novel as well as conventional thermal interface materials. It is also intended to perform reliability and thermal cycling tests on the novel materials. The test stand will aid in characterizing the impact of thermal cycling on the thermal resistance.

Major Accomplishments

The following are the main accomplishments of this project in FY 2006:

- Under a subcontract, our colleagues at the University of Colorado at Boulder achieved growth of CNTs on a copper foil. Three different growth strategies were investigated. The growth process of CNTs is being optimized in order to get the highest CNT density.
- Some preliminary thermal resistance measurements have been performed and the results are encouraging.
- A subcontract has been established with FTG to explore metal coated particles embedded in polymer compounds.
- We have specified and ordered a test stand for thermal resistance measurements. This should enable characterization of thermal resistance of the novel TIMs that are being developed as part of the project, as well as conventional TIMs, over a wide temperature and pressure range typical of automotive applications.

Technical Discussion

Importance of Thermal Interface Materials

In this section, we point out why thermal interface materials are important. Figure 1 shows the different layers constituting a typical IGBT package in an inverter. The silicon die is soldered to the direct bond copper (DBC) layer – which is composed of an aluminum nitride layer sandwiched between two copper layers. This DBC layer is attached to an aluminum base plate/heat sink via thermal grease. This thermal grease is about 25–50 microns in thickness and has a thermal conductivity of 0.8 W/mK. Typically, the inverter has a heat sink in which the coolant flows through pin fins in a channel flow configuration. Figure 3(a) shows the thermal resistance of the different solid layers in the IGBT package except for the aluminum base plate/heat sink. It shows that the grease layer has the highest thermal resistance/impedance (~ 0.1 K/W). The units are more appropriate for thermal impedance, but here thermal resistance and impedance will be used interchangeably. In order to assess the impact of thermal resistance, a 3D ANSYS conduction modeling was performed. The domain used for the simulation is shown in Figure 2 – Figure 2(a) shows the IGBT half-bridge, while Figure 2(b) shows a closeup revealing the different layers constituting the IGBT package. The details of package geometry is provided in [1].

The fluid temperature is fixed at 105°C signifying glycol-water mixture at 105°C . Except for the resistance of the interface material, all other parameter values are fixed as shown in Figure 3(a). These values are based on a typical IGBT package. As shown in Figure 3, the heat flux is fixed at 100 W/cm^2 , and the coolant temperature is 105°C and the heat transfer coefficient used on the back of the aluminum baseplate is $50,000\text{ W/m}^2\text{K}$. Figure 3(a) shows that when the resistance of the grease is about 0.1 K/W , the maximum die temperature is about 148.5°C . When the resistance of the interface material is reduced to 0.01 K/W (10 times reduction—Figure 3(b)), the die temperature reduces to 136°C . This is a very noticeable difference ($\sim 35\%$) and points to the impact of interface materials. Figure 3(c) shows that impact of the TIM thermal conductivity on the maximum die temperature. Interestingly, increasing the thermal conductivity beyond 6 W/mK does not significantly change the maximum die temperature. This implies that once the thermal resistance of the interface material is not the dominating resistance, it does not matter if its resistance is reduced further. Other layers in the package start dictating the thermal map in the package.

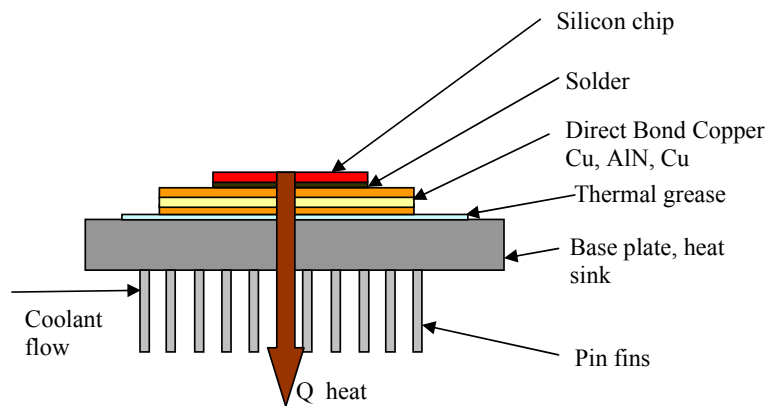


Figure 1. Different layers constituting the IGBT package in an inverter

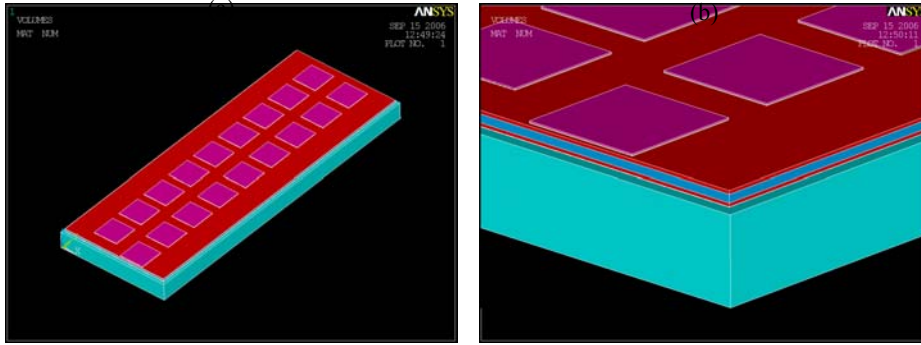


Figure 2. Domain for ANSYS 3D conduction model for illustrating the impact of TIM on the temperature fields; (a) IGBT half-bridge; (b) closeup showing part of the structure and the different layers

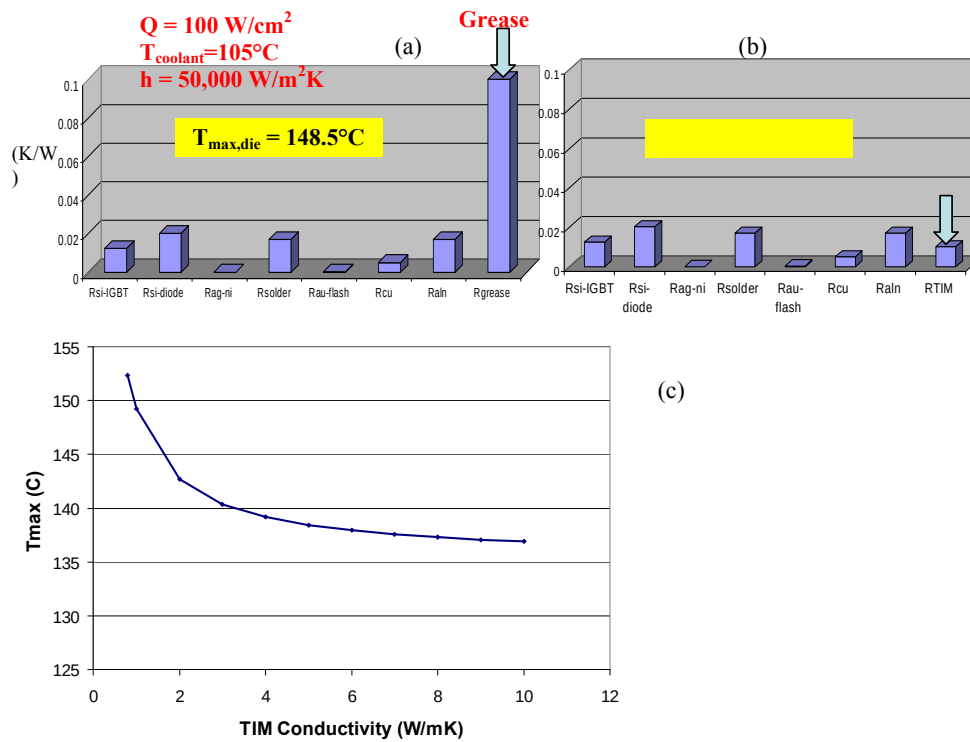


Figure 3. Impact of the thermal resistance of the TIM on the die temperature via ANSYS simulations, (a) case with typical thermal grease; (b) case with improved TIM; (c) impact of TIM thermal conductivity on the maximum temperature in the die

• State-of-the-Art of Thermal Interface Materials

In this section, we review the state-of-the-art of thermal interface materials. Figure 4 shows a schematic depiction of what a thermal interface material looks like in a real application. The surfaces of the two layers are uneven as depicted in Figure 4(a). The interface material fills up these irregularities. However, there still may be some pockets of air. Figure 4(b) shows the temperature as a function of the vertical distance. In the bulk of the material as well as in the bulk of the interface material, there is a linear temperature gradient. There is a change in the gradient at the interfaces due to the contact resistance between the interface material and the substrate. The total thermal resistance is composed of the bulk resistance and the contact resistances. Accordingly, the thermal resistance due to an interface material can be written as:

$$R = R_c + BLT/k_{TIM} \quad (1)$$

where R_c is the contact resistance, BLT is the bond-line thickness of the interface material and k_{TIM} is the bulk thermal conductivity of the interface material. A high thermal conductivity can reduce the bulk thermal resistance of the interface material; however, it is very critical that the contact resistance also be minimized. Contact resistance is an area where significant attention has been focused in the literature (e.g. [2-4]).

While the focus here is on the automotive power electronics cooling, it is worth noting that thermal interface materials also play a key role in connecting various components of the thermal solution in the microelectronics industry [4]. The continual increase in cooling demand for microprocessors has led to increased focus on improving the thermal interface materials. Figure 5 shows the state-of-the-art of thermal interface materials. Significant advances have been made in the development of thermal greases/gels, phase change materials, solder, phase change melt alloys, advanced polymers, and carbon nanotubes [4-6]. Figure 5 indicates the state-of-the-art of the best performing thermal interface materials. Also included is work from Xu and Fisher [5-7] on direct growth of nanotubes on a substrate. By growing nanotubes on both sides of the copper foil, Xu and Fisher [5, 6] have demonstrated thermal resistance of the order of $4 \text{ mm}^2\text{K/W}$ which is 5 times lower thermal resistance than the best performing greases/gels in the market.

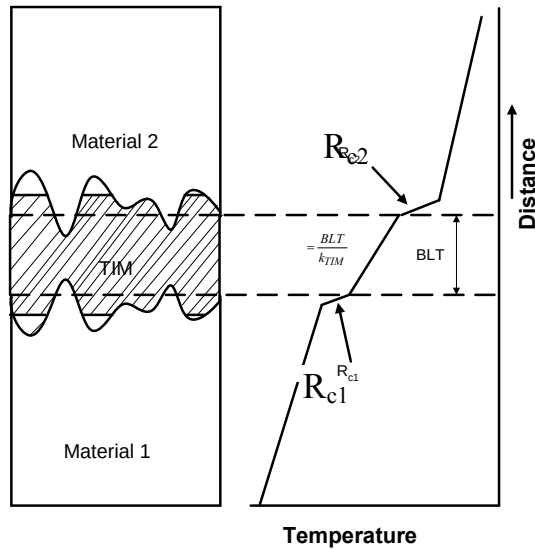


Figure 4. (a) Representation of a TIM in an actual application; (b) The depiction of temperatures as a function of distance in the vertical direction – impact of bulk resistance and the contact resistance (adapted from R. Prasher [4]).

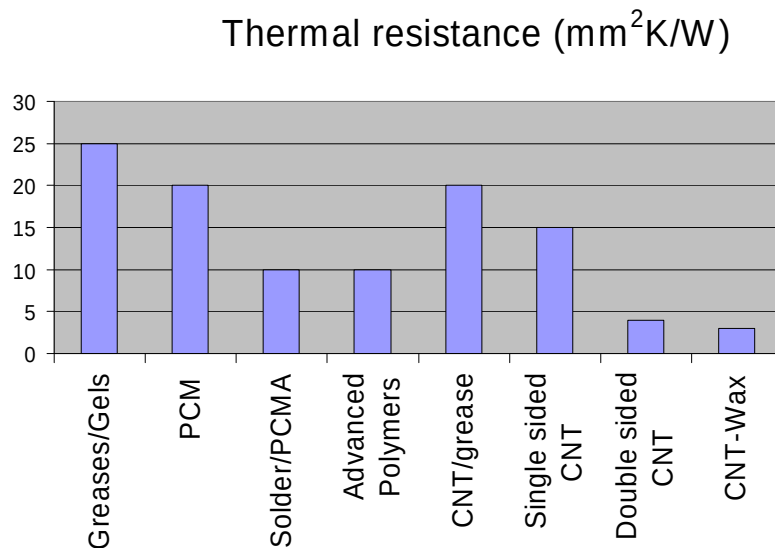


Figure 5. State-of-the-art of thermal interface materials.

Carbon Nanotubes as Thermal Interface Materials

Carbon nanotubes are currently actively being explored as thermal interface materials [5, 6, 8, 9]. Carbon-based materials (diamond and in-plane graphite) display the highest measured thermal conductivity of any known material at moderate temperatures. Their structure is indicated in Figure 6(a). In 1991, Sumio Iijima at NEC, Japan, first noticed carbon nanotubes lying in a smear of soot [10]. Carbon nanotubes (CNTs as they are commonly called) exhibit very high thermal and electrical conductivity. Hone et al. [11] showed that the thermal conductivity of single walled carbon nanotubes (SWCNTs) exceeded 200 W/mK in the direction parallel to the alignment axis of the magnetic field used in the synthesis. Kim et al. [12] measured the thermal conductivity of an individual multi-walled carbon nanotube (MWCNT) to be more than 3000 W/mK at room temperature using a microfabricated suspended device. However, theoretical predictions for the conductivity are as high as 6,600 W/mK as reported by Berber et al. [13]. As a result, CNTs offer an attractive alternative for thermal management of electronic packages.

Biercuk et al. [14] mixed 1 wt % of unpurified SWCNTs in of industrial epoxy and reported a 70% increase in the thermal conductivity of the composite at 40 K rising to a 125% augmentation at room temperature. Liu et al. [15] introduced a mixture of unpurified SWCNTs and MWCNTs into silicone elastomer and reported a 65% enhancement in the thermal conductivity of the composite with 3.8 wt % CNT loading. All these studies have relied on mixing CNTs in a grease/paste. Generally the enhancements have been of the order of 100%. It is desired to increase the thermal performance even further. Hence, researchers turned their attention to direct growth of CNTs on the substrate in the hopes of

further reducing the thermal resistance. As shown in Figure 6(b), the CNTs are directly grown on the two substrates and then the CNTs are either held together under pressure or their ends can be bonded. Hu et al. [8] characterized vertically oriented CNTs on silicon including the effect of voids and noticed the thermal conductivity to be of the order of 80 W/mK, which is one order higher than that of thermal greases, which are widely used as thermal interface materials (TIMs).

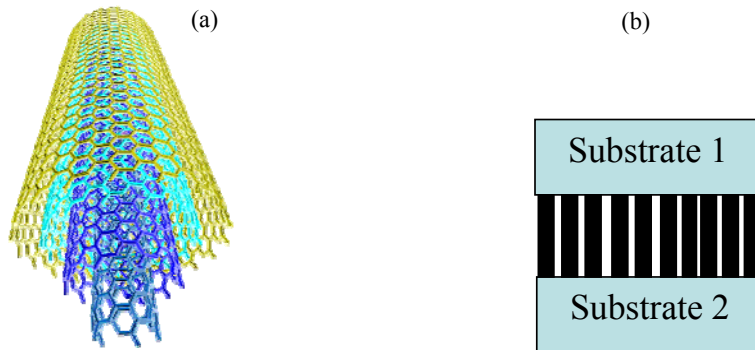


Figure 6. (a) Representation of the structure of multiwalled carbon nanotubes; (b) CNTs grown directly on substrates 1 and 2, the contact between CNTs is either a pressure contact or the CNTs could be bonded.

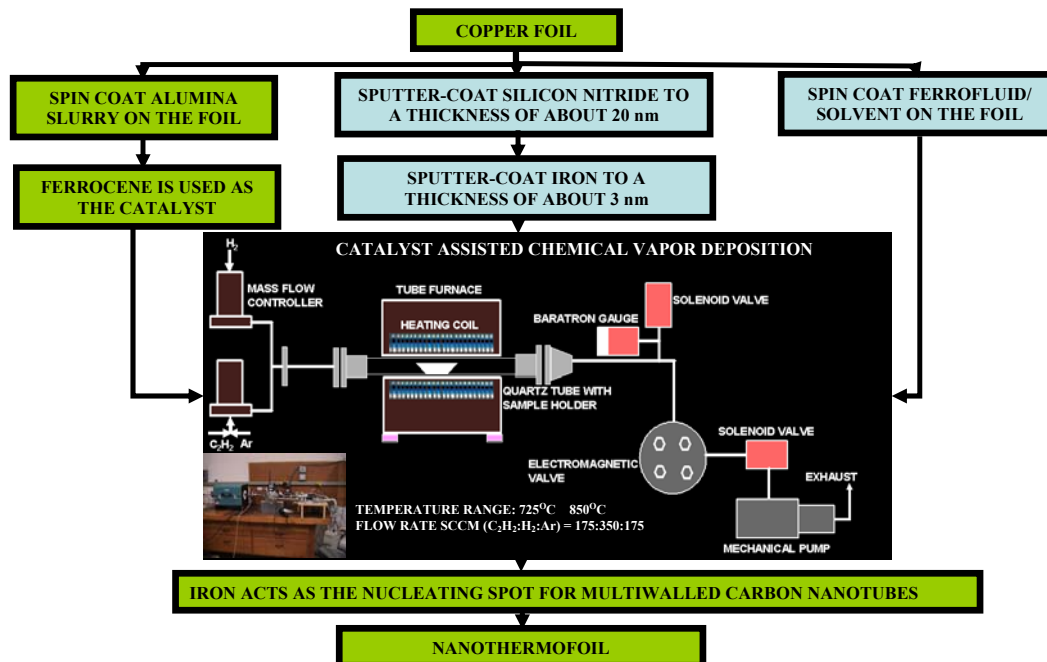


Figure 7. Summary of the growth of CNTs on a copper foil by three different methods.

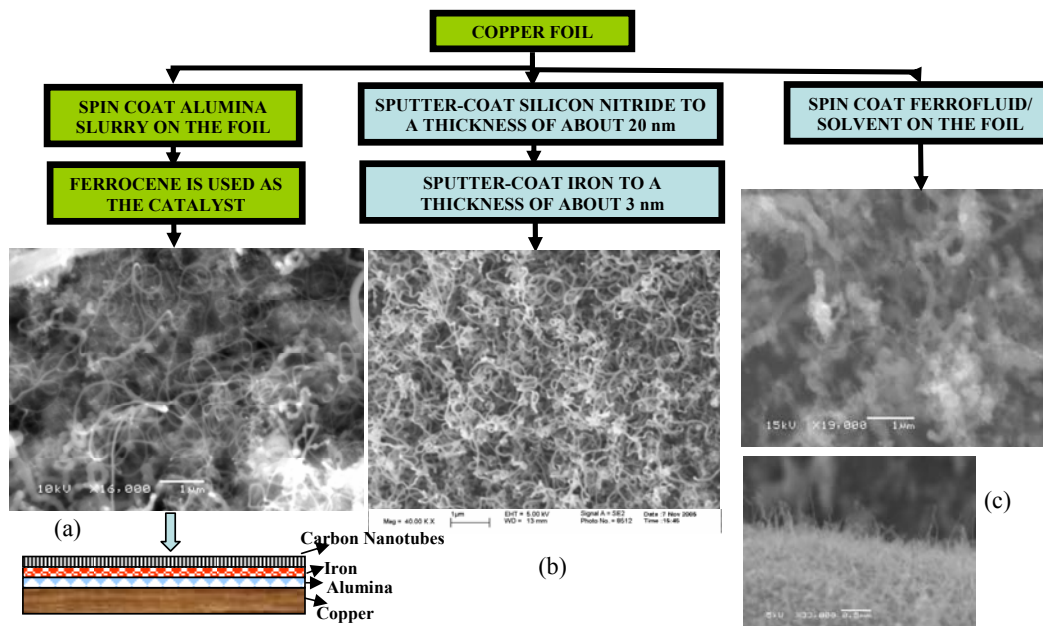


Figure 8. SEM images of growth on a copper foil via three different methods: (a) Ferrocene/alumina, (b) Silicon nitride/iron, (c) Ferrofluid/solvent.

Recently, Xu and Fisher [7] showed that the thermal contact resistance of silicon wafers and copper with a CNT array interface in between could be as low as $23 \text{ mm}^2\text{K/W}$. The same authors [6] also reported that free-standing CNT arrays can be very good thermal interface materials compared to phase change TIMs. Furthermore, they observed that a combination of PCM and CNTs produced a minimum thermal resistance of $5.2 \text{ mm}^2\text{K/W}$ at a pressure of 0.35 MPa, which is nearly five times lower thermal resistance than that of thermal grease.

Growth of Carbon Nanotubes on a Copper Foil

In FY 2006, NREL established a subcontract with the University of Colorado at Boulder to grow nanotubes on a copper foil, characterize the thermal resistance of the foil, and compare the performance with existing thermal interface materials. The growth of CNTs was achieved using a thermal chemical vapor deposition (CVD) reactor shown in Figure 7. Figure 7 outlines the overall process for growing nanotubes on a copper foil via three different methods. The intent in exploring three different methods is to ascertain which method gives the best growth density of CNTs. The assembly consists of mass flow controllers to direct the flow of the gases to the vacuum furnace that houses the sample. The mechanical pump is connected to the quartz tube to pump the reaction chamber down to a low pressure using a series of electromagnetic and solenoid valves.

Three different methods of growing nanotubes were explored as outlined below:

(a) CVD using ferrocene/alumina

Electronic-grade copper foil (99.5% purity) of thickness 1.8 mm and a mirror-polish on one side was cut into 19 mm circular blanks. The samples were sonicated in a solution of isopropyl alcohol (IPA) for 10 minutes and rinsed with distilled water to remove residual traces of the alcohol. Alumina powder with a grain size of 1 μm was cold-pressed onto the copper substrate. Then 0.15 gms of Ferrocene, the catalyst source, was taken in a quartz boat and placed near the copper sample. The furnace was then heated to 800°C after being pumped down to 0.18 Torr using a mechanical pump in the presence of Argon. A mixture of acetylene and hydrogen with a flow rate ratio of 250:495 sccm was passed over the sample for a period of 10 minutes. Once the reaction was complete, the gas supplies and the heater were switched off. The sample was then cooled in the presence of Argon. The copper foil was imaged using a Low Vacuum Scanning Electron Microscope (LV-SEM) and the result is shown in Figure 8a. The first image shows bundles of multi-walled carbon nanotubes with a wide range of diameters and chirality on a copper substrate. Along with the nanotubes, the sample also consists of the iron catalyst impurities and amorphous carbon.

(b) CVD using iron/silicon nitride film

In this process, the catalyst (iron) is laid down on the copper substrate by sputtering. Since there is a very high possibility of iron diffusing into the copper substrate at the temperatures used for processing nanotubes, a barrier layer of silicon nitride is first sputter-coated on the substrate to a thickness of 3 nm. On top of this layer, iron is sputtered to a thickness of 10 nm. The sample with the sputtered catalyst layer is placed in the furnace and heated at atmospheric pressure, to a temperature of 750°C, in the presence of nitrogen. A mixture of methane, ethylene, and hydrogen with a flow rate ratio of 50:100:500 sccm is passed over the sample for 10 minutes. Then the sample is cooled for a period of 3–4 hrs in the presence of nitrogen and imaged using a scanning electron microscope. The image (Figure 8(b)) shows entangled bundles of carbon nanotubes with small traces of catalyst impurities and amorphous carbon.

(c) CVD using ferrofluid/solvent

In this process, the sample is sonicated in a solution of isopropyl alcohol (IPA) for 10 minutes and rinsed with distilled water to remove residual traces of the alcohol. A water-based ferrofluid (EMG-707), obtained from Ferrotec, is used as the catalyst source. The ferrofluid is sonicated for a period of 10 minutes and then spin-coated on the substrate. The furnace is then heated to 750°C after being pumped down to 0.18 Torr using a mechanical pump in the presence of Argon. A mixture of acetylene and hydrogen with a flow rate ratio of 250:495 sccm is passed over the sample for a period of 10 minutes. The sample is then cooled in the presence of Argon. The image in Figure 8(c) shows nanostructures obtained using this method which are in the sub-50 nm regime. The sample also contains traces of catalyst impurities and amorphous carbon in large quantities. Investigation is underway to identify these nanostructures.

Experimental Setup for Measurement

A simple device, constructed at the University of Colorado at Boulder (CU Boulder) for measuring the thermal resistance of these copper foils with CNT growths, is schematically in Figure 9. The specimens and specimen blanks are 1 mm thick copper disks that have a diameter of 19 mm and a surface roughness of 0.13 μm . These are sandwiched between two oxygen-free, high-conductivity (OFHC) copper cylinders that are 19 mm in diameter and 24 mm long. The end of the hot-side cylinder has a heater wound on it. Three thermocouples are attached to each cylinder at known distances from the specimen interfaces. These thermocouples are set into small drilled holes and have additional wire wrapped around the cylinders for thermal anchoring. The specimens and cylinders are clamped between two aluminum plates. By using a torque wrench, specified

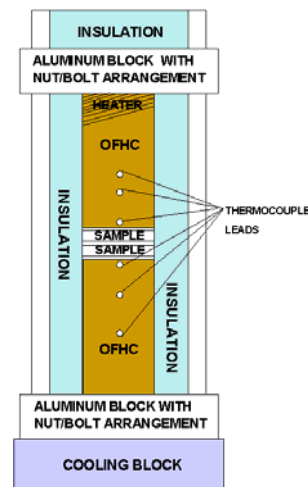


Figure 9. Schematic of experimental setup.

contact pressures can be applied to the specimens. A cooling block is bolted to the cold-end aluminum plate for heat removal, while a ceramic washer (not shown in the schematic) is inserted between the hot-side cylinder and aluminum plate. Since the cylinders and specimen blanks are copper and therefore have very high thermal conductivity, 390 W/mK, heat flows efficiently through the measurement cell.

This experimental setup is similar to that found in the ASTM E 1225 standard test method for thermal conductivity of solids by means of the guarded-comparative-longitudinal heat flow technique [16]. The primary differences between the setup used here and the ASTM E 1225 method are that this method is unguarded and the measurement of interest is the temperature difference across the specimen stack, rather than thermal conductivity. The specimen stack is surrounded, except for the cooling block, by fiber-blanket insulation to minimize heat losses. Calculation of radiative heat losses for the highest measurement power gives about 100 mW, or less than 1% radiative heat loss.

Preliminary Thermal Resistance Measurements

In this section, some preliminary experimental results for thermal resistance are reported. In Figure 10 on the upper left corner, two cases are shown: one in which there is no interface material between the copper foils, and another case in which there are CNTs between the copper foils. The plot on the left shows the temperature difference (between points A and B) vs. the power that is applied for the two cases. The case in which the CNT is between the copper foil shows 5 times smaller temperature difference as opposed to the case where there is no interface material between the copper foils. The pressure applied in the experiments is about 555 psi. The thermal resistance is also plotted in the figure on the right. It can be seen that for the case when there is no interface material between the copper foils, the thermal resistance is about 2.5 K/W, while for the case with CNTs between the copper foils, the thermal resistance drops down to about 0.5 K/W—a factor of 5 lower. This preliminary result demonstrates the impact of CNTs. In the upper right hand corner of Figure 10, the SEM image of the copper foil is shown. As

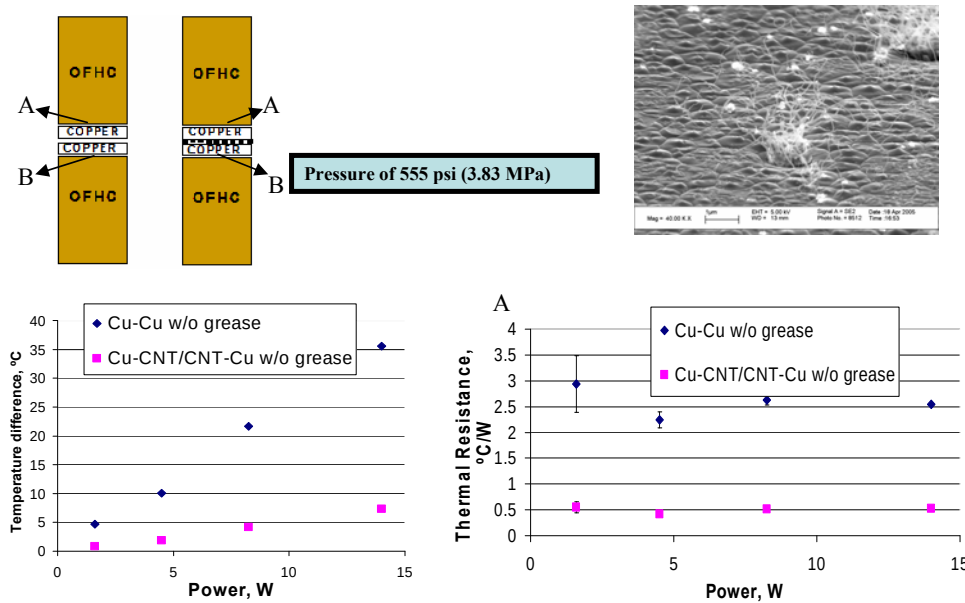


Figure 10. Experimental results demonstrating the impact of CNTs.

can be seen, the CNT growth density is very small. It is postulated that a higher density of CNTs would entail a further decrease in the thermal resistance.

Currently, at CU Boulder, additional samples are being fabricated and the thermal resistances will be characterized very soon.

- **Metal-Coated Particles Utilized in Thermal Interface Materials**

A subcontract is also about to be established with Federal Technology Group (FTG) to explore metal-coated particles embedded in polymer compounds as thermal interface materials. Coating of particles with a metal reduces the thermal resistance between the host matrix and the particles. Figure 11 shows SEM images of copper-coated diamond particles. FTG is also working with Honeywell and GE to develop these novel materials. Initially, the intent is to explore copper-coated diamond, nickel-coated diamond, and copper-nickel-coated diamond particles. Later on, other types of metal coatings and particles will also be explored. The metal coating also improves the wetting characteristics of the particles. All of this contributes towards reducing the contact resistance between the particles and the host matrix. Otherwise, even if the particle has high thermal conductivity, when it is embedded into a polymer compound, the thermal contact resistance between the particles and the host matrix hinders the overall thermal performance of the material. As with CNTs, it has been suggested that metal-coated particles could reduce the thermal resistance of polymer compounds by a factor of 5 or more.

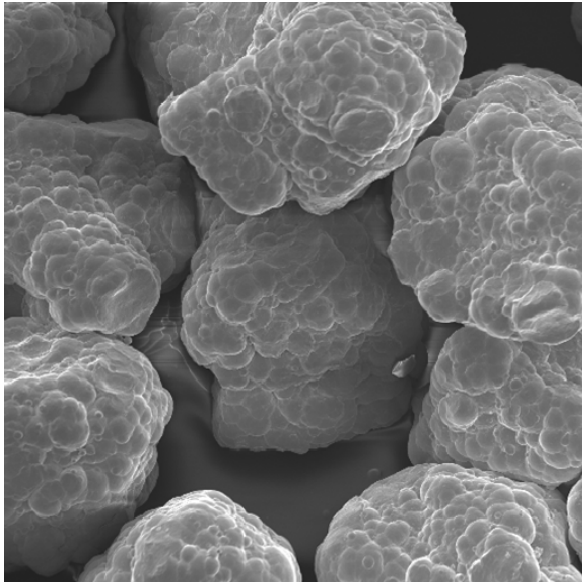
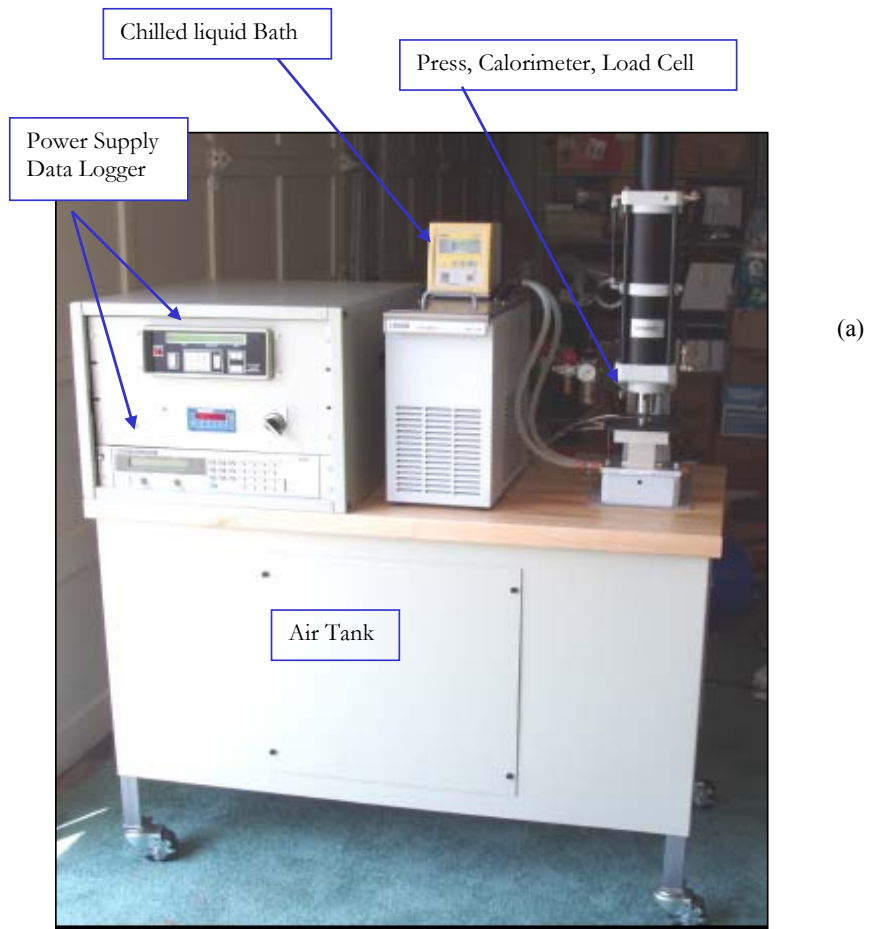


Figure 11. Copper-coated diamond particles (courtesy Federal Technology Group).

- **Test Stand for Thermal Resistance Measurements**

A test stand, based on the ASTM D 5470 test method for measuring thermal resistance, is being acquired by NREL from DS&A LLC. The test stand is expected to be delivered to NREL by the end of FY 2006. The intent of acquiring the test stand is to characterize the thermal performance of the novel interface materials being developed by the subcontractors—CU Boulder and FTG, as well other state-of-the-art thermal interface materials in the market that are suitable for automotive applications. Figure 12 shows the main components of the test apparatus. This equipment is designed to be practical and capable of applying a range of test pressures for different materials. The design for the test stand is intended to meet requirements for operation to temperatures from -40°C to $+200^{\circ}\text{C}$. The extended temperature range requires the use of a higher capacity chiller, for the cold head, and the use of a specialized silicone oil coolant.

DS&A LLC is conducting the fabrication, assembly, and calibration of TIM material test equipment which follows industry standard ASTM D 5470-01.



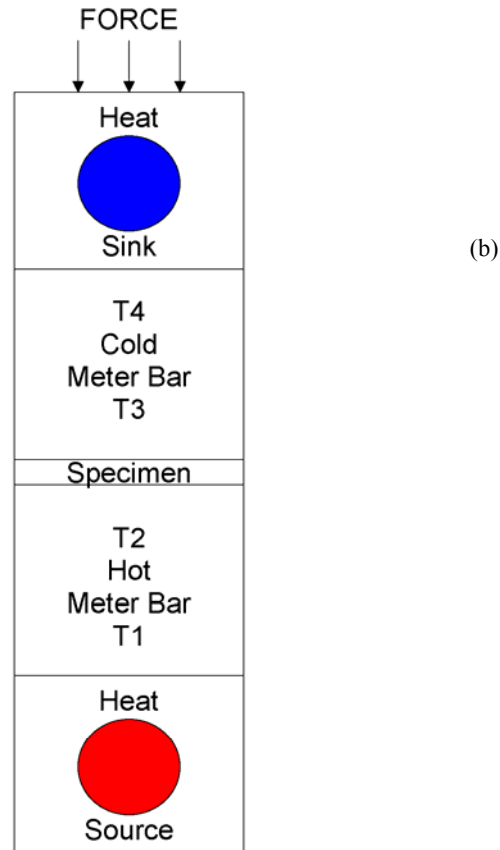


Figure 12. Test stand based on the ASTM D 5470 test method for measuring thermal resistance, (a) the complete apparatus (courtesy: DS&A LLC); (b) ASTM D 5470 measurement principle (ASTM D 5470 standard).

ASTM D 5470 Measurement Principle

The material presented below has been adapted from the ASTM D 5470 test standards document.

(a) *Heat Flow When Using the Meter Bars for Calorimeters:* Calculate the heat flow from the meter bar readings as follows (temperatures are depicted in Figure 12(b)):

$$Q_{12} = \frac{\lambda_{12} \times A}{d} \times [T_1 - T_2] \quad (2)$$

$$Q_{34} = \frac{\lambda_{34} \times A}{d} \times [T_3 - T_4] \quad (3)$$

$$Q = \frac{Q_{12} + Q_{34}}{2} \quad (4)$$

where:

Q_{12} = heat flow in hot meter bar, W,

Q_{34} = heat flow in cold meter bar, W,

Q = average heat flow through specimen, W,

λ_{12} = thermal conductivity of the hot meter bar material,
W/(m·K),

λ_{34} = thermal conductivity of the cold meter bar material,
W/(m·K),

A = area of the reference calorimeter, m²,

$T_1 - T_2$ = temperature difference between temperature sensors
of the hot meter bar, K,

$T_3 - T_4$ = temperature difference between temperature
sensors of the cold meter bar, K, and

d = distance between temperature sensors in the meter bars, m.

(b) Derive the temperature of the hot meter bar surface in contact with the specimen from the following:

$$T_H = T_2 - \frac{d_B}{d_A} (T_1 - T_2) \quad (5)$$

where:

T_H = temperature of the hot meter bar surface in contact with the specimen,
K,

T_1 = warmer temperature of the hot meter bar, K,

T_2 = cooler temperature of the hot meter bar, K,

d_A = distance between temperature sensors, m, and

d_B = distance from the lower sensor to the surface of the hot meter bar in
contact with the specimen, m.

(c) Derive the temperature of the cold meter bar surface in contact with the specimen from the following:

$$T_C = T_3 + \frac{d_D}{d_C} (T_3 - T_4) \quad (6)$$

where:

T_C = temperature of the cold meter bar surface in contact with the specimen,

K,

T_3 = warmer temperature of the cold meter bar, K,

T_4 = cooler temperature of the cold meter bar, K,

d_C = distance between temperature sensors, m, and

d_D = distance from the warmer sensor to the surface of the cold meter bar in contact with the specimen, m.

(d) Calculate the thermal resistance from

$$\theta = (T_A - T_D) \times A/Q \quad (7)$$

and express it in units of (K·m²)/W.

(e) Obtain apparent thermal conductivity from a plot of thermal resistance for single- and multiple-layered specimens against the respective specimen thickness. Plot values of the specimen thickness on the x axis and specimen thermal impedance on the y axis.

The curve is a straight line whose slope is the reciprocal of the apparent thermal conductivity. The intercept at zero thickness is the thermal interfacial resistance, R_i , specific to the sample, clamping force used, and the clamping surfaces. As a preferred alternative, compute the slope and the intercept using least mean squares or linear regression analysis.

ASTM D 5470 Model 103 Test Stand and Monitoring Equipment Fabrication, Assembly, and Calibration

- (a) ASTM D 5470 [17] is a well-defined test procedure to enable the measurement of the thermal impedance and thermal conductivity of high modulus films. In particular, this procedure is of interest in determining the performance characteristics of thermal interface materials, including films, tapes, phase change materials, and greases. Procedure modifications for the ASTM D 5470 must be incorporated for low modulus materials and for phase-change materials; these modifications are standard practice for experienced users of ASTM D 5470. For testing of phase-change materials and certain other temperature-induced materials, the test temperature should be above the phase change point of the material and physical stops must be incorporated to determine sample thickness. The hardware system proposed will allow the testing of all types of existing thermal interface materials and reveal their thermal performance characteristics for an extended temperature range. In addition, as mentioned previously, this hardware will also be utilized to characterize the thermal resistance of novel interface materials based on CNTs and metal coated particles.
- (b) The system (Figure 12) consists of the following:
- A precision air press rated at 990-lb. force at 100 psi, for two-hand operation; with pneumatic regulator. This allows adjustment of the force applied to the ram. Micrometer ram stroke is 8 in. (adjustable in 0.05 mm increments).
 - Circulating coolant bath capable of heating and chilling, temperature range -40°C to +200°C. The liquid capacity is 20 liters. Chilled coolant bath is complete with digital temperature set point and readouts. The coolant will be specialized silicone oil per recommendation of chiller manufacturer's technical service department.
 - Precision laboratory thermometer with 12 channels. Each channel can be programmed to a specific sensor. The thermometer will be furnished with four RTDs for temperature sensing. Each RTD probe is custom fabricated in a stainless steel sheath (6.0" x 1/16") with a 3-foot

cable. The thermometer will be calibrated with the probes to produce a resolution of $\pm 0.001^{\circ}\text{C}$ and an accuracy of $\pm 0.03^{\circ}\text{C}$. The data collection software is included.

- Upper and lower metering blocks (Figure 12(b)), precision machined from aluminum alloy (Al7075-0, annealed only) which has a known and documented thermal conductivity. Certifications will accompany each metering block regarding material. The sizes furnished will be one set each of the following: 1" x 1", 2" x 2".
- Agilent Technologies variable DC power supply (200 W, 120 VAC, 1.5 A) to supply power to two resistive cartridge heaters placed in the heater block above the upper metering block. Measurement of the current and voltage determines the power being applied during a test. Digital meters will visually display the volts and amps.
- A cold plate installed under the lower metering block; temperature of the cold plate is determined and regulated by (2) above. Setting this temperature determines the temperature range of the test.
- A load cell (1,000–2,000 lb) installed under the cold plate with digital readout to display the force being applied to the sample under test. The load cell will be calibrated for initial testing.
- Model 103 includes a gauge indicator to determine thickness of material under test.

Conclusion

Carbon nanotubes have been grown on a copper foil via three different techniques. The copper foil fabricated by growing CNTs directly on a copper substrate has demonstrated lower temperature drops across the interface when compared to plain copper substrates. Initial experiments reveal that a factor of 5 improvement was achieved with CNTs incorporated at the copper interface when compared to plain copper substrates at all power levels. More rigorous testing is currently being performed. Metal-coated particles are also being explored for use in thermal interface materials. A test stand is being acquired to characterize the thermal performance of the novel as well as conventional TIMs over temperature and pressure range suitable for automotive applications.

Next Steps

The following activities are planned for FY 2007 and beyond:

- More detailed thermal resistance measurements will be performed to characterize the performance of CNTs over a range of temperature and pressures. Their performance will be compared against the best possible commercially available thermal interface materials.
- Via a subcontract to FTG, metal-coated particles embedded in polymer compounds will be explored.
- The performance of metal-coated particles embedded in polymer compound will also be characterized via thermal resistance measurements over a range of temperature and pressure. Comparisons will be made with commercially available interface materials.
- Typically carbon nanotubes can be grown by chemical vapor deposition process. Our collaborators at CU Boulder intend to explore a spraying technique for depositing nanotubes on the copper foil which would eliminate the chemical vapor deposition process.
- The possibility of replacing the grease in the IGBT package with these novel TIMs will be explored. This may involve interaction with OEMs and suppliers to figure out all the practical aspects.
- The cost impacts of using CNTs and metal coated particles will be examined more closely.

Publications

“Fabrication And Thermal Characterization Of Copper-Carbon Nanotube ‘Nano-thermofoil’ As An Interface Material,” to be presented at IMECE 2006, ASME International Mechanical Engineering Congress and Exposition, Nov. 5-10, 2006, Chicago, IL.

References

- [1] Narumanchi, S.V.J., Hassani, V. and Bharathan, D., 2005, “Modeling single phase and boiling liquid jet impingement cooling in power electronics,” National Renewable Energy Laboratory, NREL Report No. TP-540-38787. www.nrel.gov/docs/fy06osti/38787.pdf.
- [2] Madhusudana, C.V., 1996, “Thermal contact conductance,” New York, Springer-Verlag.
- [3] Yovanovich, M.M. and Marotta, E.E., 2003, “Thermal spreading and contact resistances,” in Heat Transfer Handbook, A. Bejan and A.D. Kraus, Editors, John Wiley & Sons, Hoboken, NJ, pp. 261–395.
- [4] Prasher, R., 2006, “Thermal interface materials: historical perspective, status and future directions,” to appear in the Proceedings of the IEEE.
- [5] Xu, J. and Fisher, T., 2006, “Carbon nanotube array thermal interfaces,” Cooling Technologies Research Center, Purdue University, CTRC Report # 2006-01.
- [6] Xu, J. and Fisher, T.S., 2006, “Enhancement of thermal interface materials with carbon nanotube arrays,” International Journal of Heat and Mass Transfer, 49, pp. 1658–1666.
- [7] Xu, J. and Fisher, T.S., 2004, “Enhanced thermal contact conductance using carbon nanotube arrays,” Proceedings of the ITherm conference: ninth intersociety conference of thermal and thermomechanical phenomena in electronic systems, Las Vegas, NV.
- [8] Hu, X., Padilla, A.A., Xu, J., Fisher, T.S. and Goodson, K.E., 2005, “Thermal characterization of vertically-oriented carbon nanotubes on silicon,” Proceedings of the 21st IEEE SemiTherm symposium, San Jose, CA.
- [9] Tong, T., Majumdar, A., Zhao, Y., Kashani, A., Delzeit, L. and Meyyappan, M., 2006, “Indium assisted multiwalled carbon nanotube array thermal interface material,” IEEE SemiTherm conference, Dallas, TX.
- [10] Iijima, S., 1991, “Helical microtubules of a graphite carbon,” Nature, 354, pp. 56–58.
- [11] Hone, J., Llaguno, M.C. and Nemes, N.M., 2000, “Electrical and thermal transport properties of magnetically aligned single wall carbon nanotube films,” Applied Physics Letters, 77(5), pp. 666–668.
- [12] Kim, P., Shi, L., Majumdar, A. and Mceuen, P.L., 2001, “Thermal transport measurements of individual multiwalled nanotubes,” Physical Review Letters, 87(21), pp. 215502(1–4).
- [13] Berber, S., Kwon, Y.-K., and Tomanek, D., 2000, “Unusually high thermal conductivity of carbon nanotubes,” Physical Review Letters, 84(20), pp. 4613–4616.
- [14] Biercuk, M.J., Llaguno, M.C., Radosavljevic, M., Hyun, J.K., Johnson, A.T. and Fischer, J.E., 2002, “Carbon nanotube composites for thermal management,” Applied Physics Letters, 80(15), pp. 2767–2769.
- [15] Liu, C.H., Huang, H., Wu, Y. and Fan, S.S., 2004, “Thermal conductivity improvement of silicone elastomer with carbon nanotube loading,” Applied Physics Letters, 84(21), pp. 4248–4250.
- [16] ASTM, 1986, “Annual Book of ASTM Standards,” Vol. 14.02, E 1225, Philadelphia.
- [17] ASTM. *ASTM D5470-01 Standard*. 2005.

Nomenclature

<i>A</i>	Area
ASTM	American Society of Testing and Materials
BLT	Bond line thickness
CNTs	Carbon nanotubes

CVD	Chemical vapor deposition
DBC	Direct bonded copper
IGBT	Insulated gate bipolar transistor
k	Thermal conductivity
LVSEM	Low vacuum scanning electron microscopy
OFHC	Oxygen-free high conductivity copper
PCM	Phase change materials
PCMA	Phase change metallic alloys
Q	Heat flow
R	Thermal resistance/impedance
RTD	Resistance temperature detector
T	Temperature
TIM	Thermal interface materials

Greek

λ	Thermal conductivity
θ	Thermal resistance