Abstract:

The Most Common High Power RF Design Error: Inadequate Back-Side Grounding of the Source Contact Connection

Freescale's Application Support Department often assists design engineers in troubleshooting their newly created high power RF amplifier designs. A wide number of RF performance issues are due to a very common design mistake.

By a wide margin, the number one cause for most RF performance issues can be traced back to a serious shortcoming in the device's back side RF grounding connection and its electrical contact with the associated PCB ground plane.

Text Body:

The Most Common High Power RF Design Error: Inadequate Back-Side Grounding of the Source Contact Connection

On a daily basis, the Freescale Application Support Department receives numerous calls from RF design engineers who are in crises mode. Usually, these design engineers have created a design using the best simulation tools possible but after building their first prototype units, they experience RF performance issues of varying degrees. Sometimes the problem is poor RF performance numbers that are not in line with the data sheet. On other occasions, the problem relates to spurious oscillations, or shifting RF performance numbers with temperature. In extreme cases, the problem is associated with total device failures.

By a wide margin, the number one cause for most RF performance issues can be traced back to a serious shortcoming in the device's back side RF grounding connection and its electrical contact with the associated PCB ground plane.

There are two key assembly requirements that must be accomplished by the devices' back side interface.

The interface must provide a good thermal contact between the RF device and heatsink. This interface needs to be flat, smooth, and include a microscopic gap filing interface material like solder, thermal grease or a thin elastomeric pad of some sort. All of these attributes are needed in order to facilitate the heat flow out of the device and into the heatsink-to-air assembly.

Some common mistakes made here include excessive solder voiding, excessive thermal grease thickness and excessively rough machined surfaces with no gap filling material.

The second item that must be created in the back side interface is a solid, consistent RF planer ground. Not a DC point ground but a broad, ultra low electrical resistance connection between the back side of the RF device and the back side of the PCB. Notice that this is not the heatsink. The heatsink may be used as part of the grounding structure or it may not, but the key required function is to mesh the PCB ground plane and the RF device's ground plane with a low resistance, high-current carrying assembly method.

The most common mistakes that occur with regard to the PCB ground plane/RF device interface are associated with the PCB grounding. At low frequencies and low RF powers, mount the PCB to the heatsink using screws which creates point contact DC grounds. However, this does not work at higher frequencies above 500 MHz and higher powers above 100W. One needs to solder the back side ground of the PCB to the back side ground of the RF device using a copper carrier or heat spreader assembly in order to be able to handle the high RF circulating currents that are created in this area.

At RF power levels above 100W and with low RF impedances at the device's leads (near 1 ohm), the RF currents tend to be large;
P= I**2/R or I=10 Amps in the given example. Designers take great pains to ensure the top side PCB impedance matching structure is consistent, robust, repeatable. The device drain lead interface to the PCB trace must also possess an ultra low RF resistance connection.

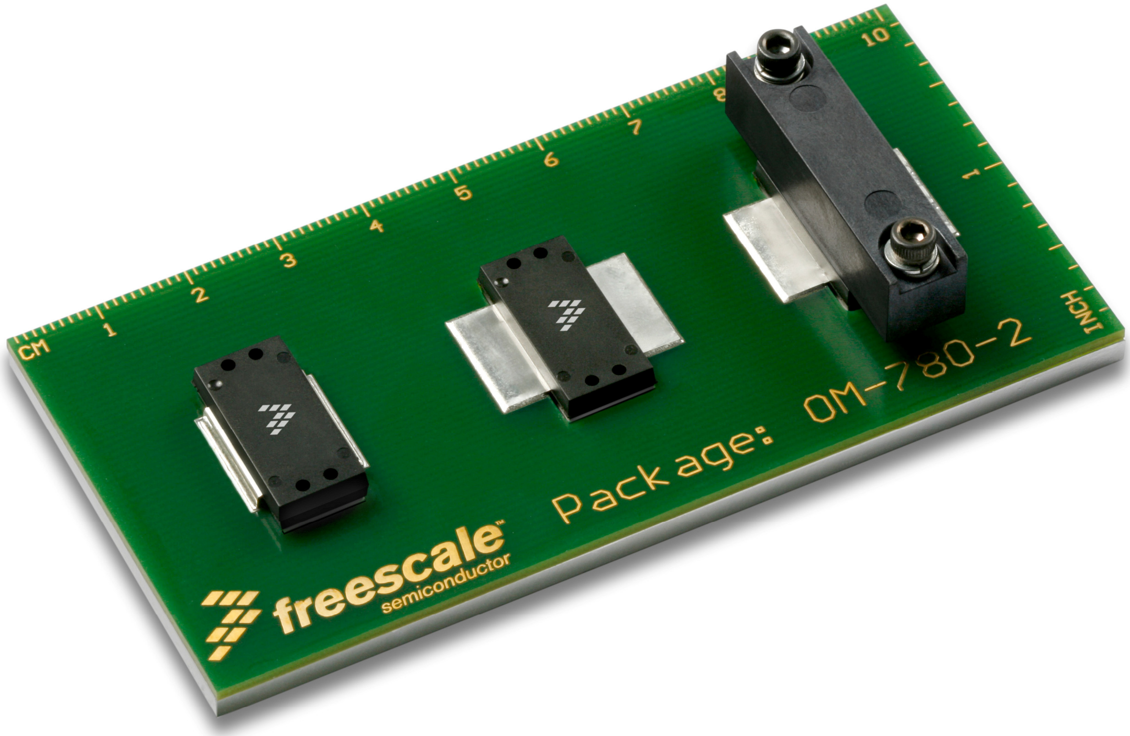
Just as there are 10 amps of RF current flowing on the top side of the PCB, then there also must be 10 amps flowing on the back side ground plain between the PCB and the RF device. Yet that interface structure is often largely uncontrolled. The back side ground consistency is just as important as the top side matching consistency and yet the back side ground is never documented, rarely measured and very often overlooked. To remedy this, solder the back side assembly with the same accurate and consistent solder process used on the top side assembly process.

Another common mistake relating to the PCB ground plane/RF device interface is the application of a chemical chromate finish to the aluminum heatsink to prevent oxidation. Class 3 is the insulating version and Class A1 is considered electrically conducting but the reality is that both prevent the ultra low contact resistance that is required in a back side RF grounding application. That is why the heatsink is a poor choice in providing the electrical connection between the PCB and the RF part. A much better design methodology is to utilize a consistent, robust, fully soldered assembly that controls both the top side matching and the back side return path thru a carrier, coin, heat spreader or PCB back side metal layer.

Conclusions

Clearly a fully soldered assembly in the critical area between the back side source of the device and the PCB ground plane through a heat spreader is the optimum design solution. Anything less and one is asking for trouble.

For reference, here is a list of mounting application notes that are available from the freescale.com/rfpower web site: AN1908, AN1940, AN2467, AN3778 and AN3789



Picture 1, Three methods of mounting a RF power Device. From the left, True Surface Mount Soldered with Gull-Wing Leads, Back Side Solder Down, and Screws with a Center-Pushing Clamp.