

# GENERAL APPLICATION NOTE

## QPI FAMILY

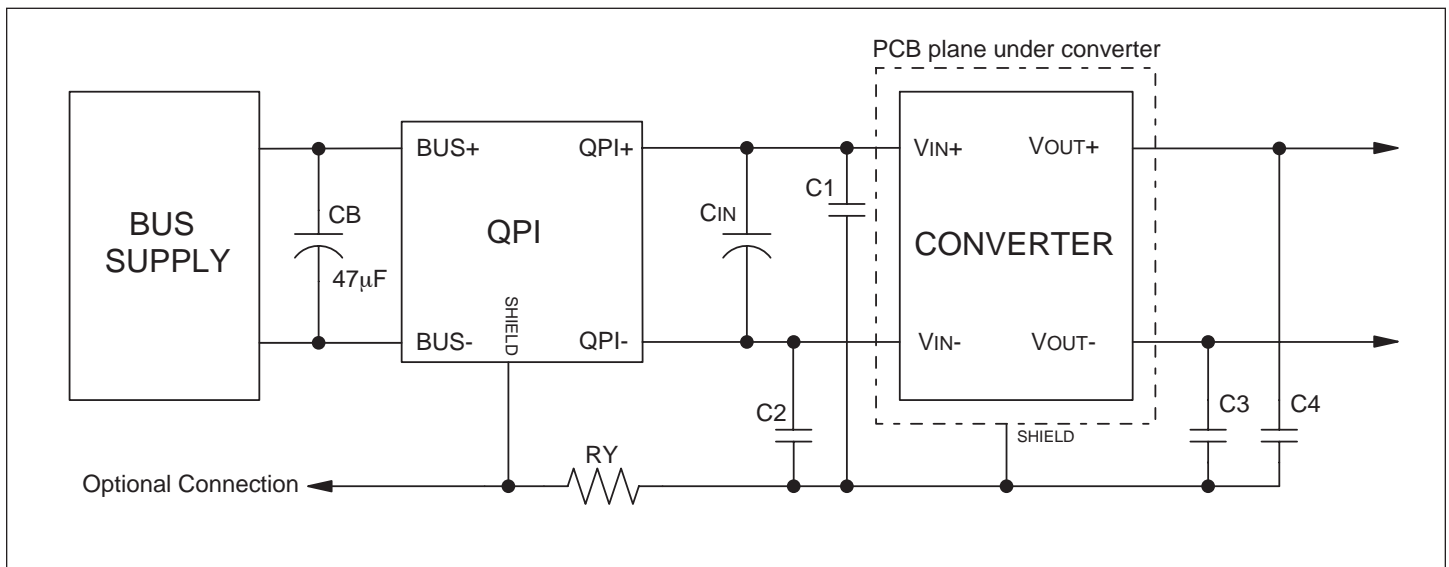
EMI control is a complex design task that is highly dependent on many design elements. Like passive filters, active filters for conducted noise require careful layout and attention to grounding to ensure highest performance. This application note will describe how to use the QPI to solve noise problems with DC-DC converters. Recommended PCB layouts and suggestions for external components are made to help the user obtain the best performance of the QPI family. Measurement techniques and system level suggestions are also included.

Designers should be aware that to select and qualify an EMI filter for conducted noise, they must test the filter in their product under the set up and conditions specified in the applicable EMI standards. Filter selection or design must be based on the pre-filtered noise magnitudes and the frequency spectrum of concern. A product’s conducted noise profile includes differential and common mode noise. It may also include radiated noise depending on the EUT shielding and cabling screening in the measurement setup. The

IEC (International Electrotechnical Commission) special committee on radio interference specification CISPR 16-2-1 describes the methods of measurement for conducted disturbances.

Filter performance in the application is highly dependent on the input bus and load impedance and cannot be extrapolated from zero bias 50 Ohm insertion loss data alone. Final noise performance is a complex function of filter elements, equipment grounding and noise source impedances, which vary in magnitude and phase over the frequency spectrum of interest. Several before and after examples are included to show the performance of the QPI. Measurements are made with the test set up shown in *Figure 5* and several representative DC-DC converters. Results are shown in the Appendix.

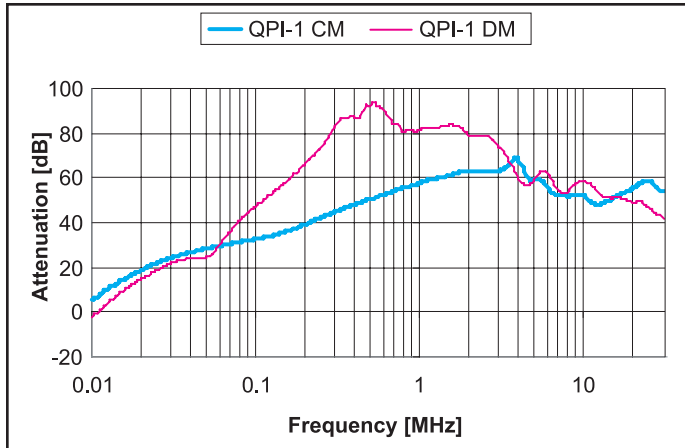
QPI evaluation boards are available which include a mounted QPI with screw terminals to allow easy in-circuit testing. Check [picorpower.com](http://picorpower.com) for data sheets and evaluation board details.



**Figure 1**—Recommended schematic for QPI. Values of  $C_{IN}$  and  $C1-C4$  should be the values recommended by the converter manufacturer. If the manufacturer does not specify a value for the Y-capacitors use 4.7nF.  $R_y=10\ \Omega$ , 1/4 W. **Read important note on page 3 for Layout and Circuit Considerations.**

## QPI Operation

The QPI products are EMI filters providing active common mode attenuation of conducted noise over the EN55022 range from 150 kHz to 30 MHz. The QPI senses the common mode current flowing in the bus lines and creates a low impedance at the shield plane pins 5 & 6 to re-circulate the noise back to the generating source. When connected as shown in *Figure 1*, the control loop will actively drive the shield pin and reduce the common mode current in the bus lines to values approaching the common mode current ratio attenuation curve in *Figure 2*.



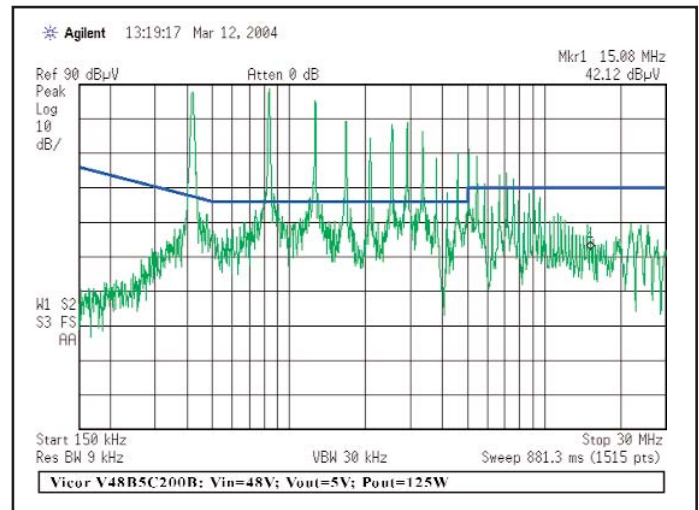
**Figure 2** - Insertion loss curves showing common mode and differential mode attenuation.

*Figures 3 and 4* show the effect of inserting the QPI-1 filter on a QPI-1-EVAL1 board between the DC bus and the converter input under the test set up shown in *Figure 5*. The resulting plot in *Figure 4* shows that the QPI-1 is effective in reducing the total noise spectrum measured pre-filter to well below the EN55022 Class B Quasi-peak detection limit.

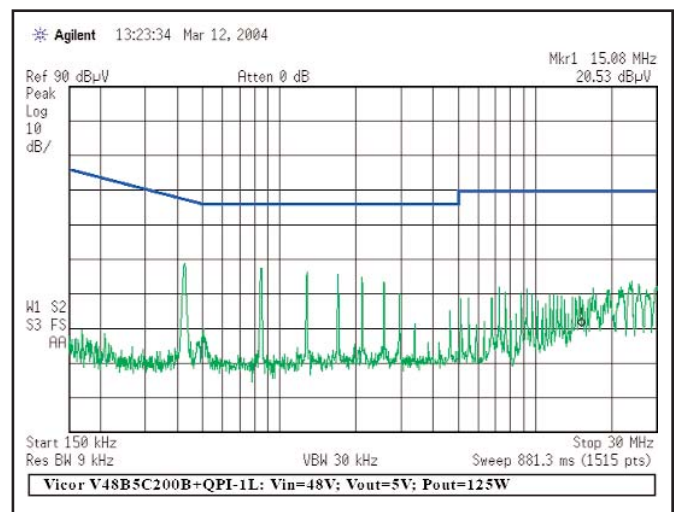
The Appendix shows several examples using the QPI-1 with industry standard converters with before and after plots of the noise voltage spectral profiles under the recommended test set up.

## Noise and Sources

The first step in designing to meet conducted emissions standards is to minimize the noise generators. The conducted noise sources measured on the supply bus and reference ground within a system are most often caused by the power conversion section and, to a lesser degree, other sources such as data I/O lines or PWM fan motor driver circuits.



**Figure 3** – Conducted EMI profile of a DC-DC converter without filter.



**Figure 4** – Conducted EMI profile of a DC-DC converter with QPI.

With brick type converters, certain topologies and manufacturers generate less noise than others. Knowledge of the converter's base-line noise is critical. The next step is to minimize the noise sources' coupling paths to the power sources. Most manufacturers of DC-DC converters recommend decoupling their products with passive components and shielding via PCB ground planes. Even the best low noise converters usually require additional filtering and decoupling to pass compliance testing, especially the EN55022 Class B limit.

The two components of conducted noise are differential and common mode. **Differential mode** noise is created on the power bus by the AC component of input differential current caused by the changing input current the converter presents to the bus supply that provides the input power. The differential mode current creates what is defined as symmetrical voltage in CISPR16-2-1.

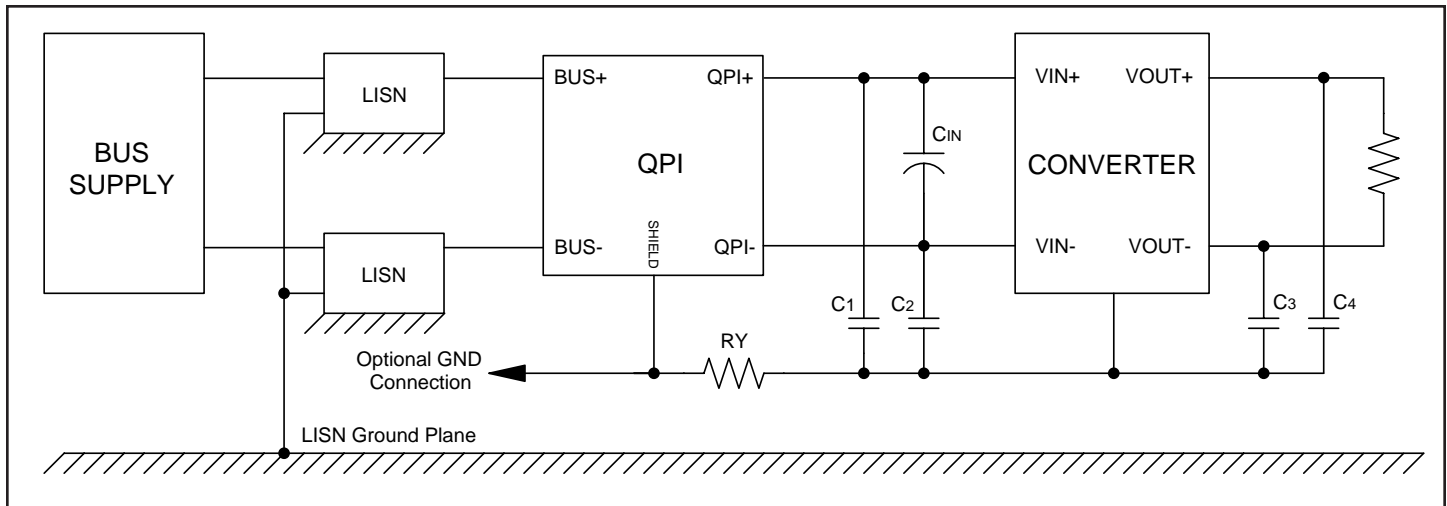


Figure 5—Spectrum Analyzer Test Setup (QPI and converter 5 mm above LISN Ground Plane)

Most converter manufacturers specify a minimum input capacitance ( $C_{in}$ ) to provide a low input source impedance for stable operation. Recommended large-value electrolytic capacitors do not have low impedance at the harmonic frequencies of the converter fundamental. Therefore, low value capacitors with good high frequency characteristics should be connected in parallel with the electrolytic input capacitor(s). Even with this approach, some differential AC current will flow from the bus, causing differential mode noise currents in the bus lines.

To meet the lowest level of the Class B specification, the differential current in the conducted EMI frequency range should be less than 21 dB $\mu$ A. This equates to 55 dB $\mu$ V or 562  $\mu$ V of rms voltage across 50 ohms. The length of bus lines can vary depending on system installation. To mitigate the effects of this variable, it is recommended to use CB (see Figure 1) to provide a low source impedance to the QPI. This minimizes input excursions during transient loads reflected from the output of the converters to the QPI. The bus line parasitic inductance and CB form additional filtering for differential noise current.

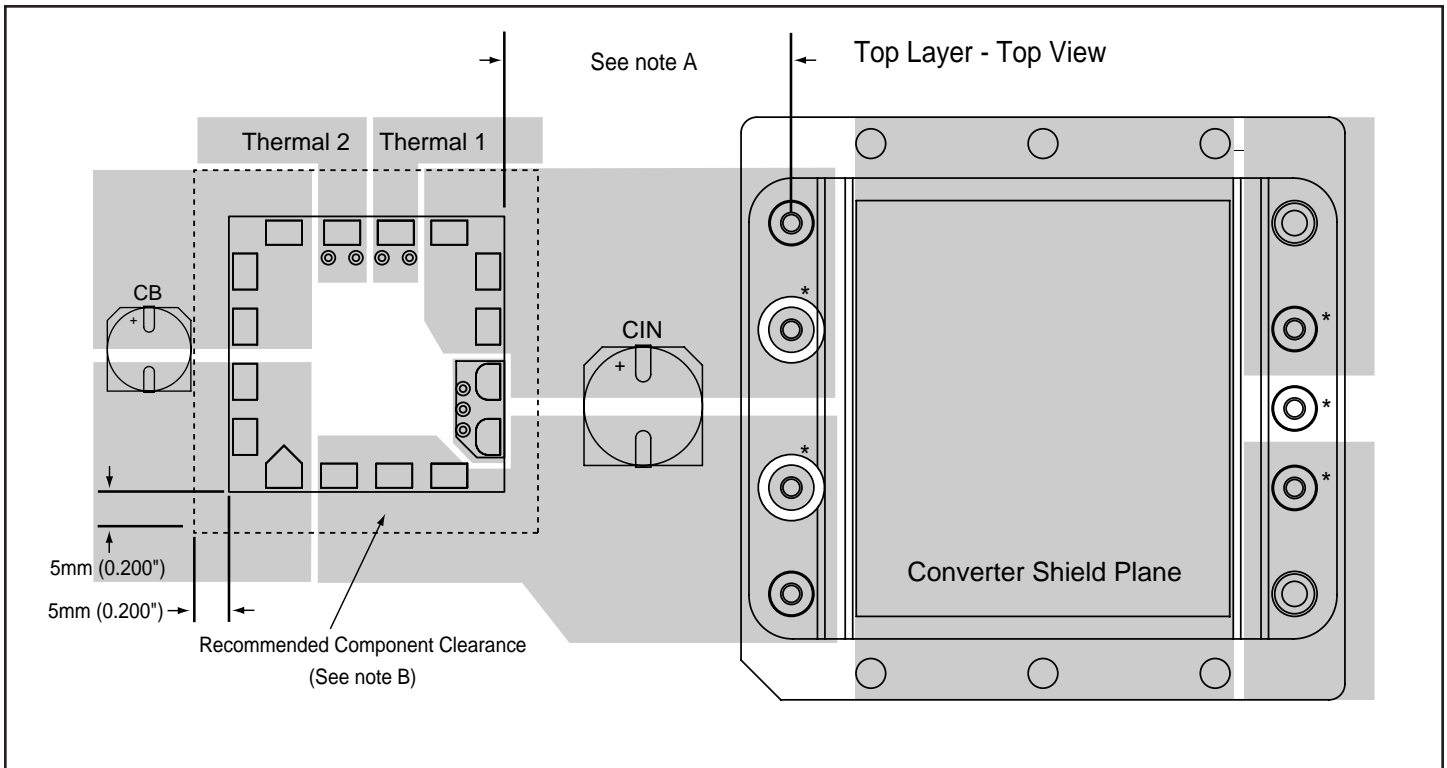
**Common mode** noise on the power bus is defined as as unsymmetrical voltage in CISPR 16-2-1. CISPR 16-2-1 also describes common mode disturbances as those primarily measured for signal and control lines. The QPI is designed to solve common mode noise problems on the power bus.

Common mode noise is caused by high  $dV/dt$  E fields, usually at the primary switch, secondary rectifiers and power transformer parasitic capacitance. B fields of the power magnetics and parasitic inductances caused by high levels of  $dI/dt$  in the converters are also common

mode noise sources. All noise currents generated by these high fields must return to their source. Best practice for the E fields is to provide a return path with capacitors through a common shield ground plane as shown in Figures 6 and 7 ( $C_1$ - $C_4$  are usually referred to as "Y" caps).

**Important Note: For optimum performance of the QPI with most converters, it is critical to connect the QPI shield plane (pins 5 & 6) to the converter EMI shield plane (common point of  $C_1$ - $C_4$  "Y" capacitors) thru RY as shown in Figure 1 and layout Figure 7.  $C_{in}$  and  $C_1$ - $C_4$  should be the values recommended by the converter manufacturer. Absent a recommendation, use 4.7 nF for  $C_1$ - $C_4$ . If the EMI ground return connection (chassis or case) is used in the power conversion section of the PCB, it should be connected at the QPI shield plane, shown as the optional connection in Figure 1. Connecting the converter EMI shield plane directly to chassis safety ground may degrade the QPI attenuation performance.**

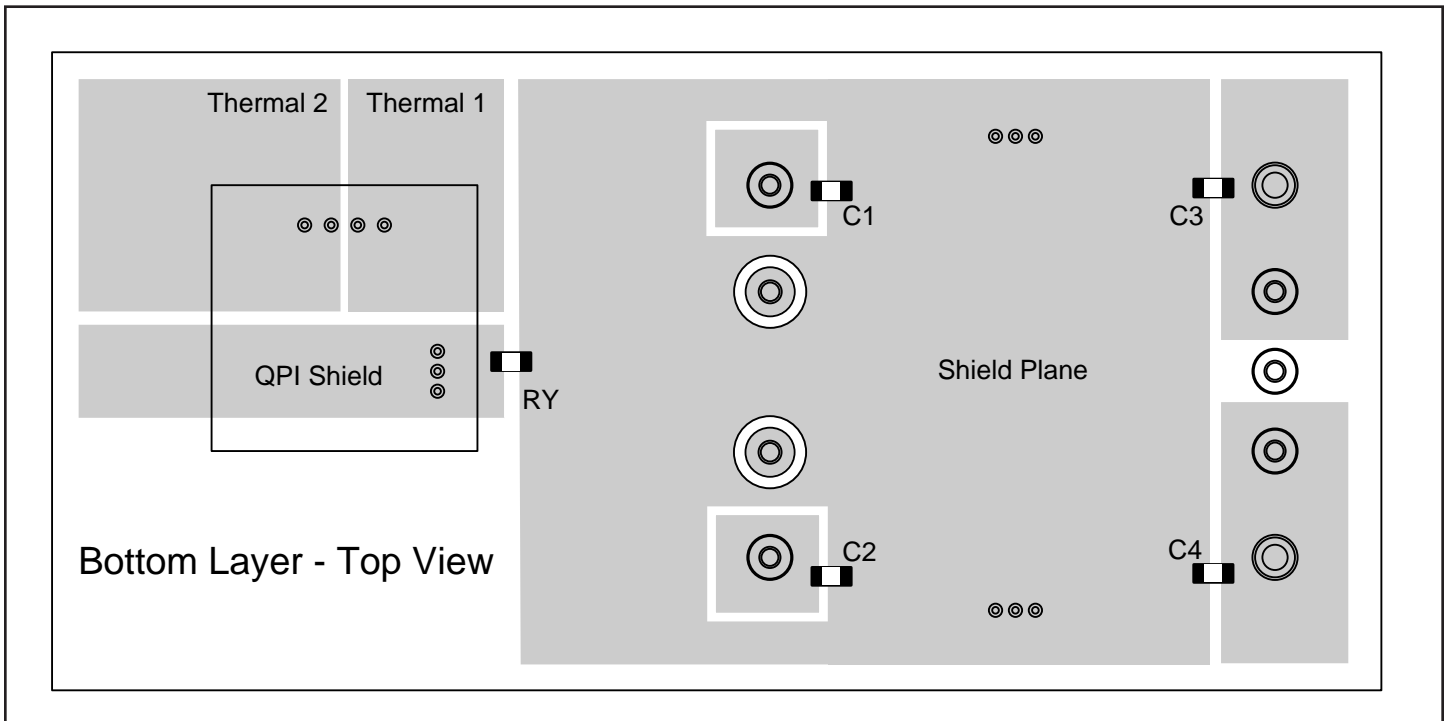
**Some systems may require the converter positive or negative input or an output "terminal" to be connected to PE (Protective Earth) ground for safety or other considerations. When using the QPI in this situation this "terminal" must be connected to the converter shield plane created in the PCB layout under the converter. Because the PE earth path may pass excessive current under a fault condition the resistance of this path may be limited to a low resistance value. To meet the resistance requirement without degrading filter performance RY can be replaced by a 4.7uH inductor rated for the fault current condition maintaining low power dissipation during a fault until the protection device clears.**



**Figure 6** – Recommended layout for top layer of PCB.

**Note A:** Some converters may radiate energy past the QPI into the input bus degrading noise reduction of the filter. To minimize the potential for this effect use a maximum feasible distance between the QPI and the converter in the layout. The input filter and any bulk hold up capacitors should be placed in this area.

**Note B:** Hako de-soldering nozzle model no. A1137 and appropriated temperature controller can be used for removing and re-soldering SiP packages. Clearance around the SiP is a keep out zone for components as noted by the dashed line to use this tool. Components taller than 10mm (0.400") require 15mm (0.600") clearance from the SiP package edge.



**Figure 7** – Recommended layout for bottom layer of PCB. Note location of RY.

---

The QPI is a surface mount product designed to be used with PCB mounted open frame and baseplated converters, but can also be designed into embedded power supply designs. If present, the baseplate/heat sink, normally isolated from all converter pins, acts as a coupling plate to the high fields created in the converter and should be tied to the PCB shield plane via conductive mounting hardware as shown in *Figure 1*. Some converters have a case pin that should be connected to the PCB shield plane.

The QPI family of products with 48 Vdc inputs, are Class I safety rated devices providing basic insulation from input and output to the shield plane pins. The voltages in telecom systems are considered TNV-2 which per UL 60950 3rd edition, Clause 3.6 are treated as non-hazardous potentials requiring only basic insulation.

For a Class I system or product, there is parasitic inductance in the safety earth ground connection from the chassis of the system (EUT) to the earth ground point. CISPR 16-2-1 delineates the effects of grounding on the conducted noise measurements using AMNs in Annex A3 and A4. Under CISPR 16-2-1 recommendations for noise measurement, the noise profiles presented were made with the EUT ground connected to the AMN/LISN reference ground with the same length of ordinary cable as the bus lines, approximately 1 meter.

Some systems may have the converter baseplate(s) mounted to the chassis for heat sinking. If a system is constructed with grounded base plates, care must be taken when making noise measurements. If the chassis is set on the reference ground plane during the noise measurement, noise will couple from the chassis to the ground plane, skewing the test results. Measuring noise with a chassis shorted directly to the LISN reference ground is in conflict with the recommendations of CISPR 16-2-1. It is recommended that users follow CISPR and tie the chassis ground back to the reference ground at the AMN/LISN with 1 meter of cable.

Common mode noise of open frame converters is not zero and will couple through any parasitic capacitive path to the chassis or case of the system and may cause EMI compliance failure. Conduction occurs via the safety return path, so it is critical that the shield plane and "Y" capacitors be incorporated as shown in *Figure 7* to create a local low impedance return path, minimizing common mode current flowing back through the power source.

The common mode noise current that does not return to it's source via the shield plane and "Y" capacitors will,

as previously stated, couple to any adjacent case or chassis plane capacitance that will be connected to a safety ground. The noise current will flow back to it's source through the low ac source impedance of the bus+ and bus- paths. The system earth ground wire will ultimately connect to earth ground at a power panel or an earth grounding common location. It is the safety ground connection that enables conducted common mode current to flow even though it is not galvanically tied to the noise source generators. The purpose of the filter, as with a discrete common mode choke, is to create a high series impedance path back to the power source and/or provide a low impedance return path for the noise current back to the respective noise source. The active filtering within the QPI works in this manner.

### Compliance Testing

For compliance testing, the ground plane of the Line Impedance Stabilization Network (LISN) provides the return path. All tests in this note were performed using 50Ω/50μH Solar Electronics Type 8028-50-TS-24BNC LISN as shown in the test circuit in *Figure 5*. In compliance testing, the common mode currents flow through the LISN's 50 Ohm measurement paths of the bus+ and bus-, returning to their source in the equipment under test (EUT) via the wires connecting the LISN to the EUT.

The CISPR22 test limits are for total noise over the specified spectrum for a particular system. Either differential or common mode noise components can cause compliance failure and generally both have to be attenuated to pass the Class B limit. The class distinction details will not be covered in this note but essentially Class A permits higher noise levels, typically in fixed systems, versus Class B for general consumer and portable applications. The standard for fixed systems has recently been raised to the Class B conducted noise limit in the AdvancedTCA™ PICMG® 3.0 requirements for individual data communications boards.

There are also different detection techniques that affect the limits used for the pass/fail criteria. The EN55022 Quasi-peak detection Class B limit is the limit line in all plots in this note. The results shown in this document were obtained with the test configuration shown in *Figure 5*. The peak detection option of the spectrum analyzer was used instead of quasi-peak to dramatically reduce the sweep time of the analyzer. Using the Quasi-peak detection measurement mode would result in lower measured amplitudes by the factor this detection method introduces.

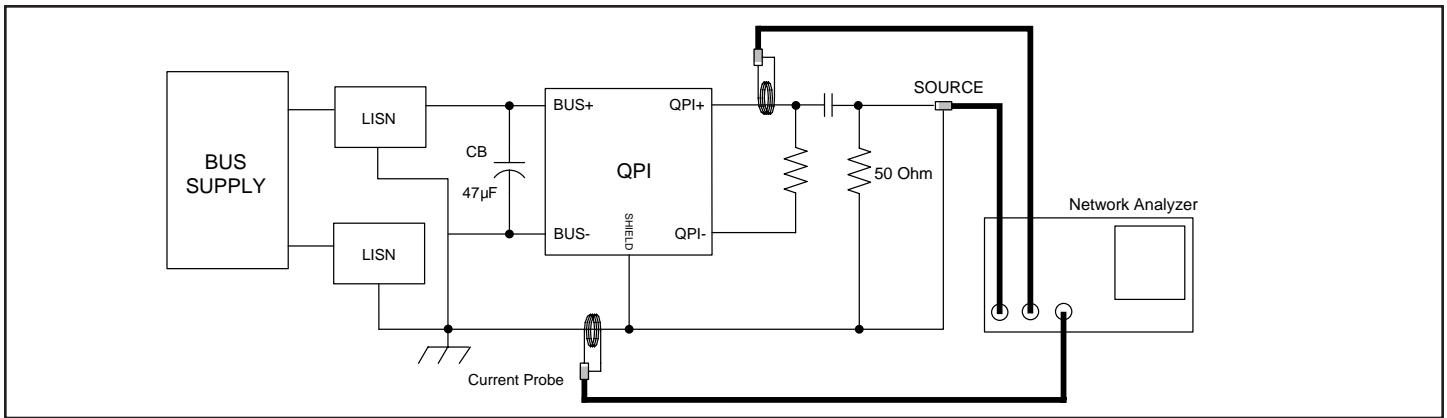


Figure 8—Common Mode Attenuation Measurement Setup

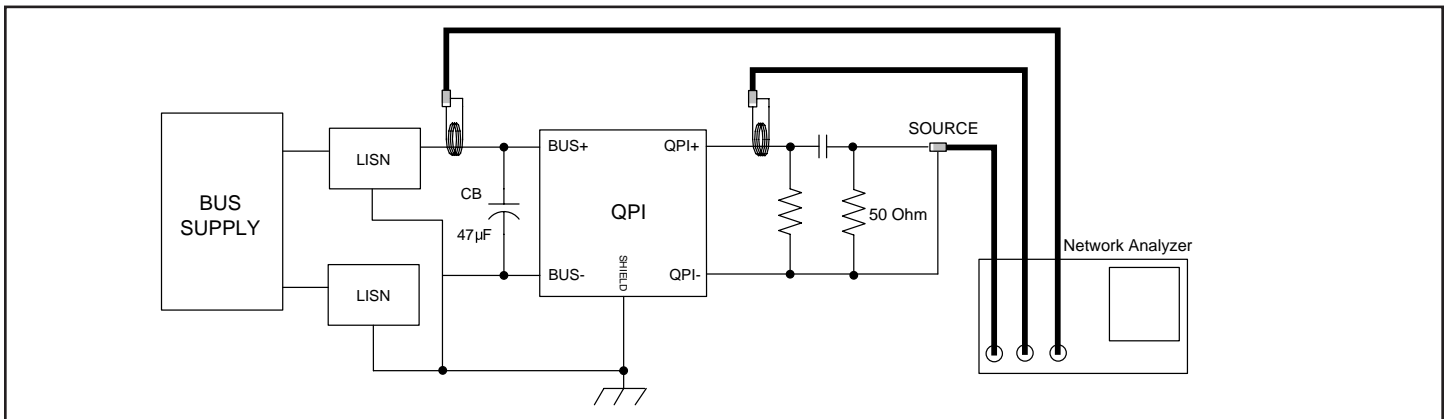


Figure 9—Differential Mode Attenuation Measurement Setup

## QPI Characterization

Two methods of measurement were used to provide the QPI characterization data in this note. The first method was to measure the small signal sinusoidal current ratios of injected input current to filtered output current with a network analyzer per CISPR 16-2-1. The data is presented as an attenuation ratio in decibels versus frequency. A nominal bias was applied for all measurements due to the nature of the product. The second method was to measure absolute noise voltage levels using a spectrum analyzer with operating converters before and after insertion of the QPI filter. All tests were performed with AMNs (Artificial Mains

Networks) as defined in CISPR 16-2-1 for the frequency range of 150 kHz to 30 MHz. The AMNs are also referred to as LISNs (Line Impedance Stabilization Networks).

The attenuation curves in the data sheets and the example in *Figure 2* were generated with a network analyzer signal source coupled into the differential and common mode paths separately. They show the ratio of output to input current in dB over the conducted frequency band. The test circuits in *Figures 8* and *9* were used to make the measurements. The attenuation curve is measured with a small signal sinusoidal source under conditions replicating the actual application test.

It measures the ratio of injected rms current to the bus rms current and is analogous to 50 Ohm insertion loss testing used to characterize passive filter solutions. Ultimately, the QPI must be tested in the system to determine if it will bring a product or system into compliance.

## Layout Considerations

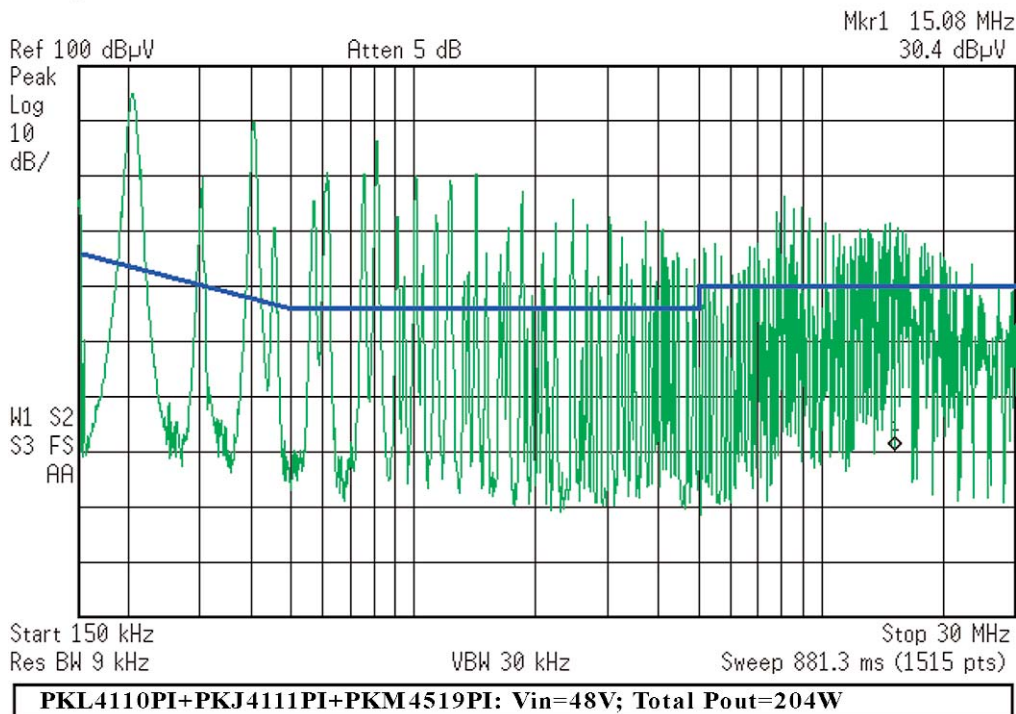
Dissipation of the QPI must be considered when laying out a board to use the QPI. The Therm 1 and Therm 2 pins provide a thermal path to remove heat from the QPI and they should not be electrically connected to any other points. To achieve the junction to case ratings in the data sheet, the pad areas on a two sided PCB should be equal to the size in the recommended layout footprint shown in *Figures 6* and *7*. The area for each of these lands should be approximately 1.0 in<sup>2</sup> of 1 ounce copper. Thermal vias should be used to provide a thermally conductive path to the bottom layer for the Therm 1 and Therm 2 copper lands. If the PCB temperature will exceed 60°C under full system power load and air flow, the current derating curves for the respective products should be observed. See the product data sheets for temperature derating curves. Use 1.3 mm minimum spacing between the shield pads and the safety ground path to adjacent lands to maintain the hi-pot rating.

Appendix of additional converters tested with no filtering then with a QPI-1EVAL1 board.

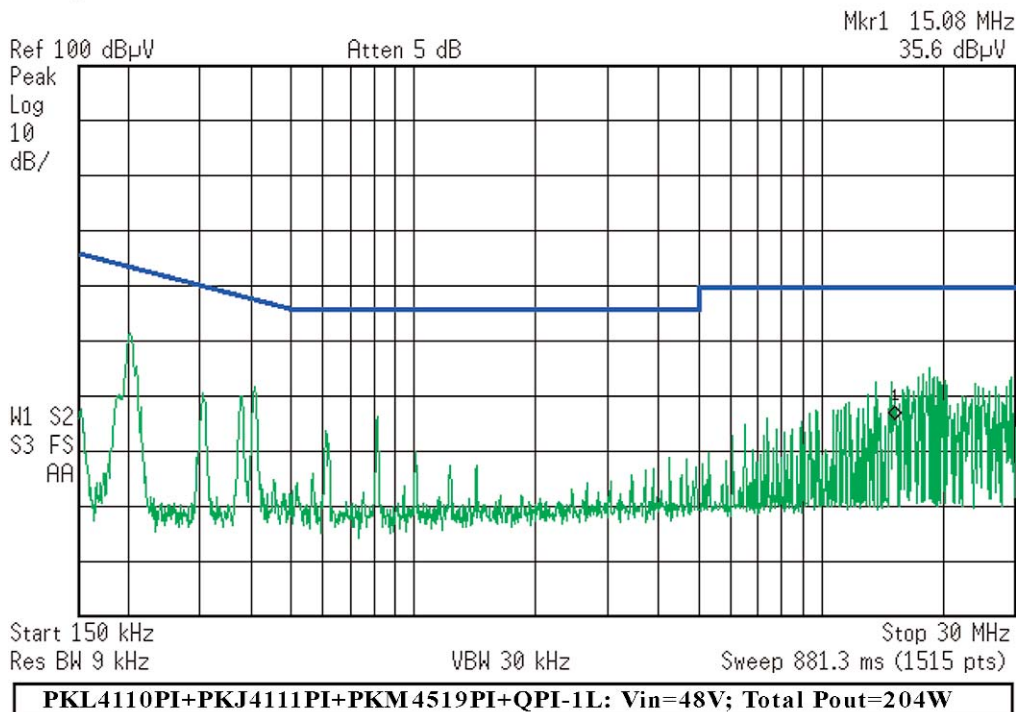
Multiple Ericsson Converters:

| Converter | Output Voltage | Iout Max | Switching Frequency | Applied Load |      |
|-----------|----------------|----------|---------------------|--------------|------|
| PKL4110PI | 3.3 V          | 50 A     | 150 kHz             | 29.7 A       | 98 W |
| PKJ4111PI | 5.0 V          | 20 A     | 200 kHz             | 15.0 A       | 75 W |
| PKM4519PI | 2.5 V          | 20 A     | 150 kHz             | 12.5 A       | 31 W |

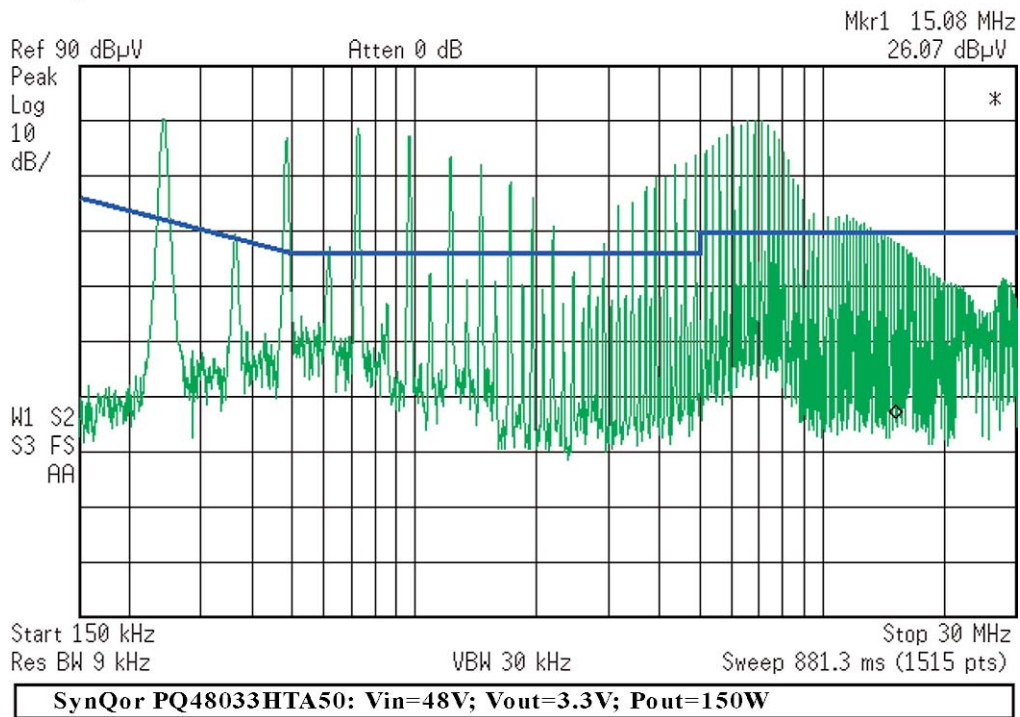
\* Agilent 13:44:04 Mar 12, 2004



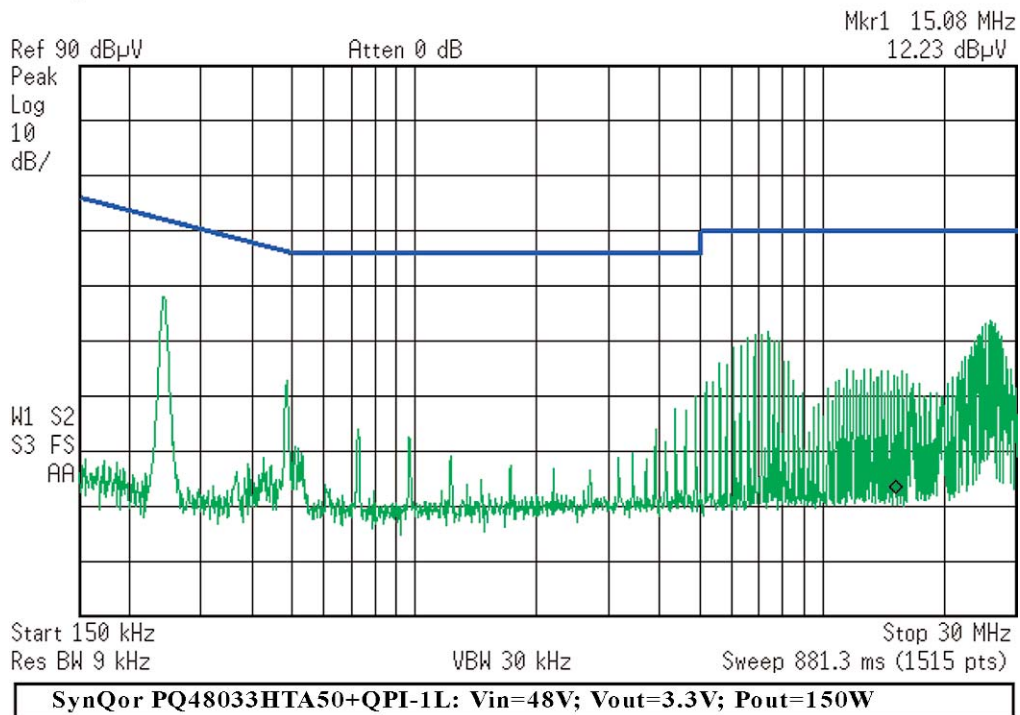
\* Agilent 13:50:30 Mar 12, 2004



\* Agilent 14:24:34 Mar 12, 2004

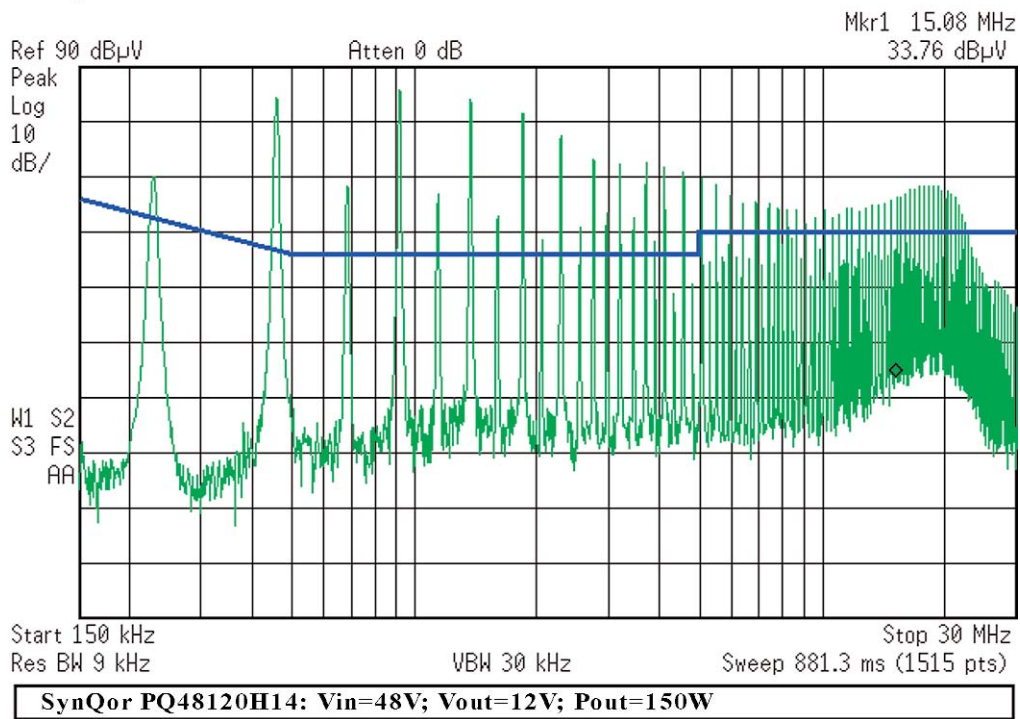


\* Agilent 14:29:33 Mar 12, 2004

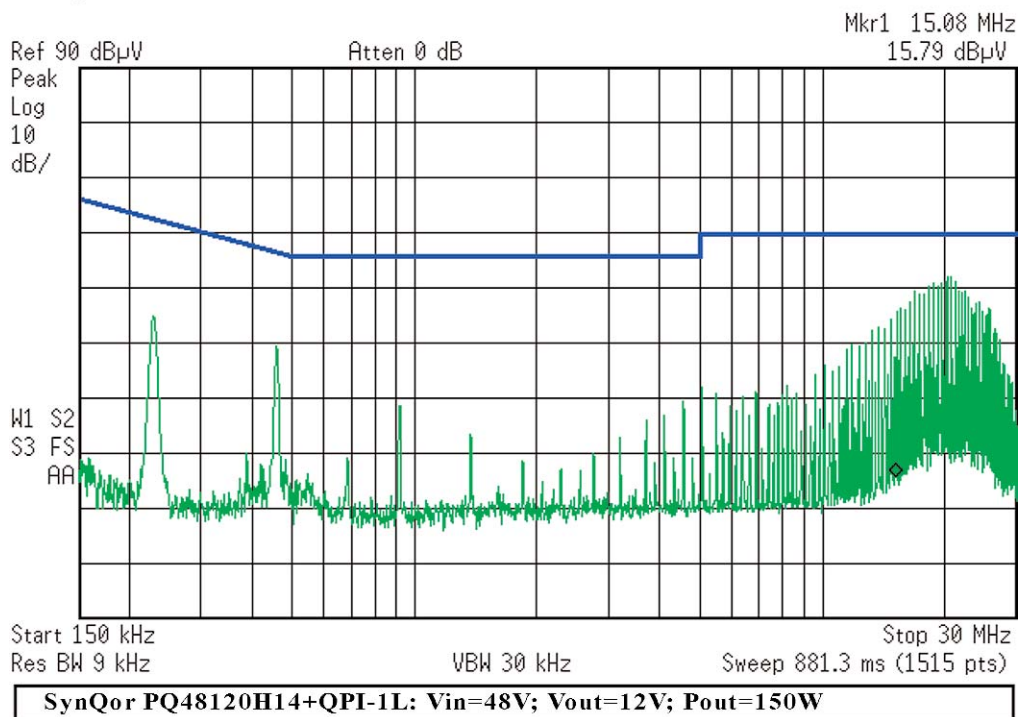




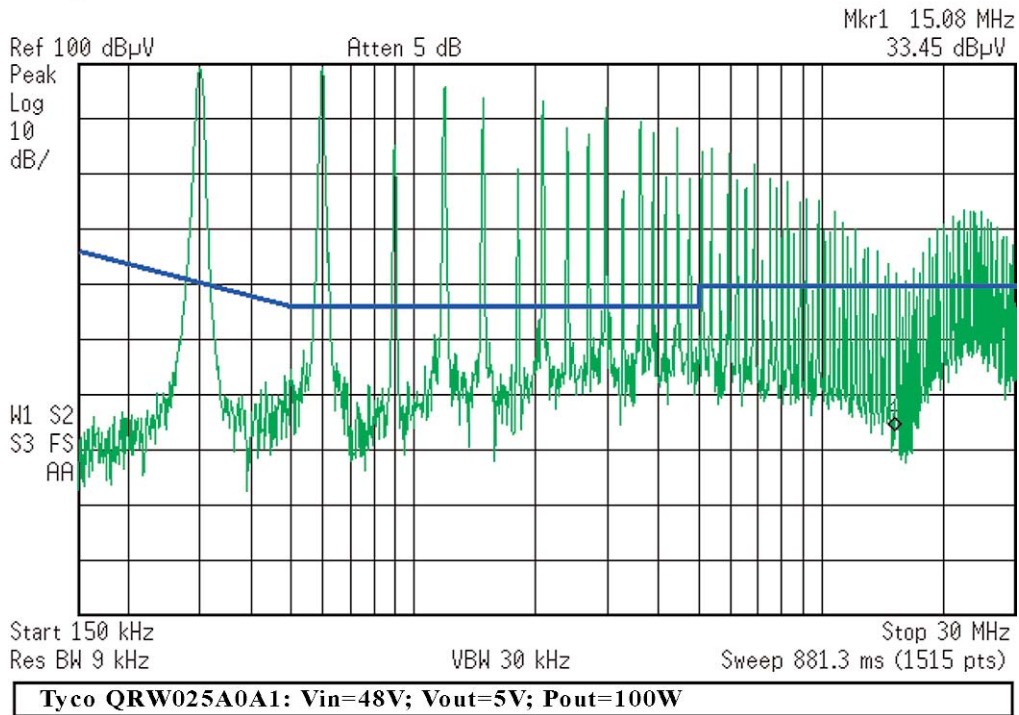
Agilent 15:09:14 Mar 12, 2004



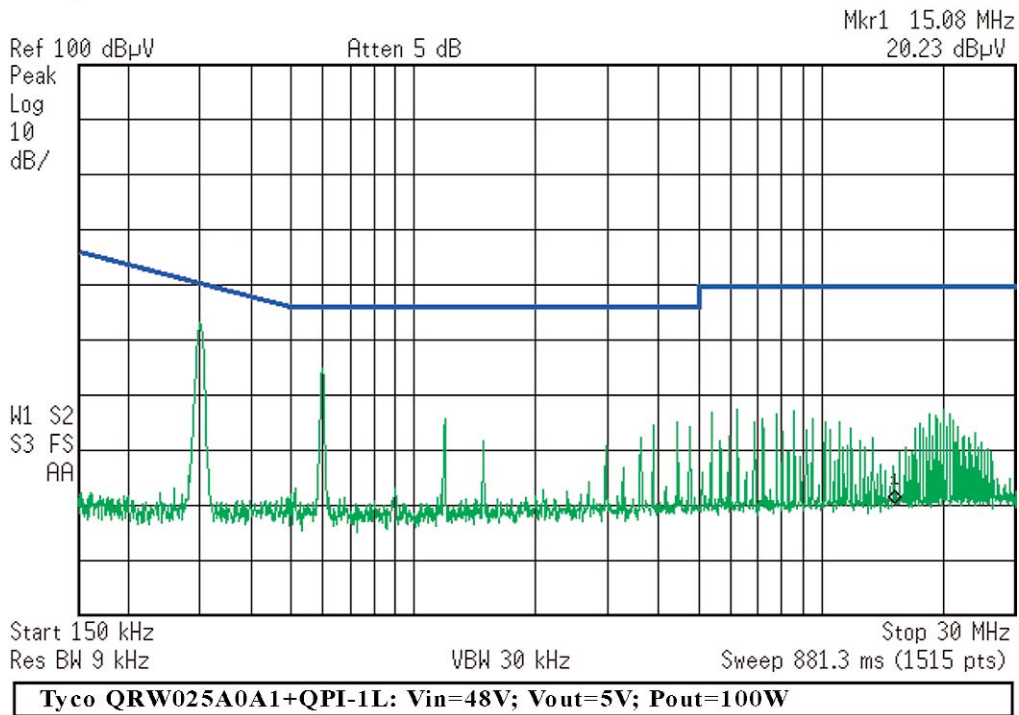
Agilent 15:05:02 Mar 12, 2004



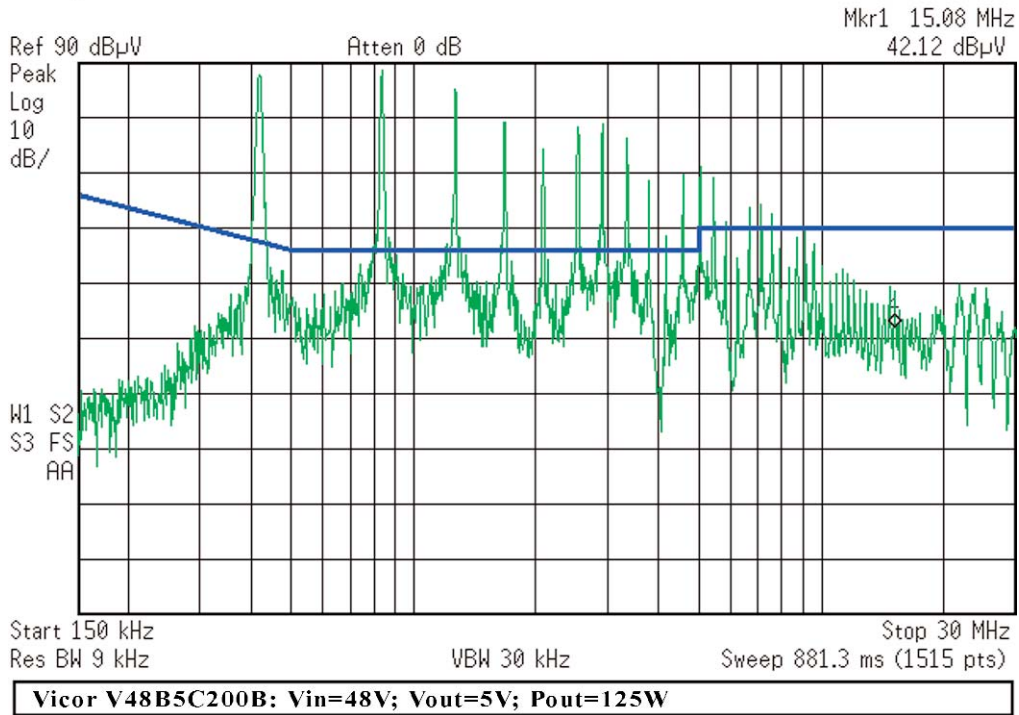
Agilent 15:20:43 Mar 12, 2004



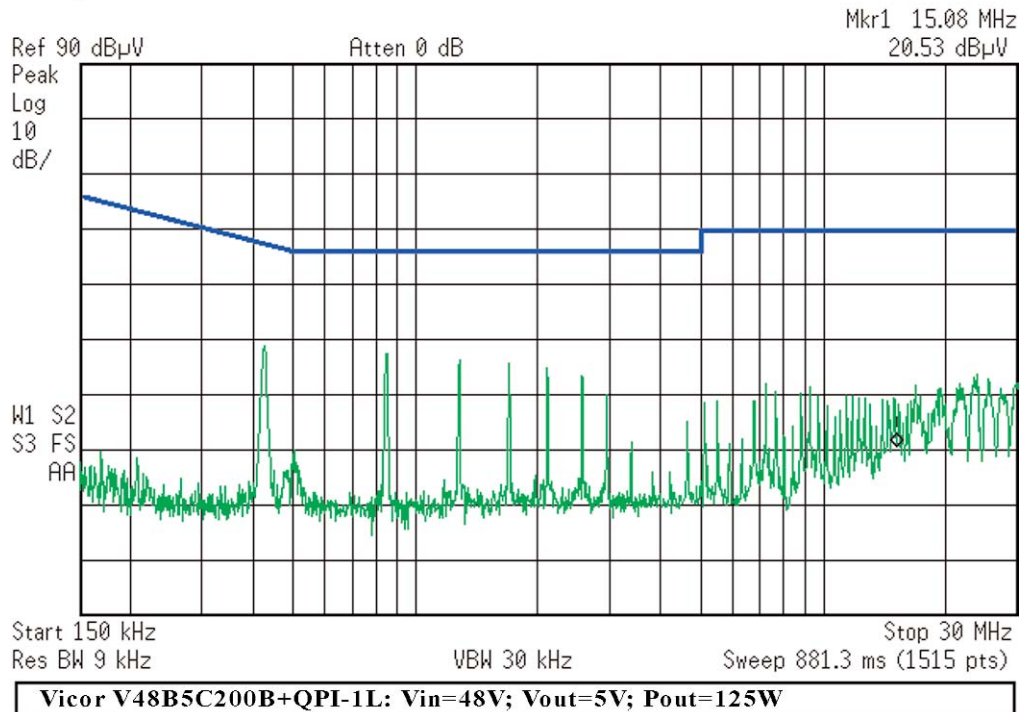
Agilent 15:23:48 Mar 12, 2004



Agilent 13:19:17 Mar 12, 2004



Agilent 13:23:34 Mar 12, 2004



---

**Vicor's comprehensive line of power solutions includes high-density AC-DC & DC-DC modules and accessory components, fully configurable AC-DC & DC-DC power supplies, and complete custom power systems.**

Information furnished by Vicor is believed to be accurate and reliable. However, no responsibility is assumed by Vicor for its use. No license is granted by implication or otherwise under any patent or patent rights of Vicor. Vicor components are not designed to be used in applications, such as life support systems, wherein a failure or malfunction could result in injury or death. All sales are subject to Vicor's Terms and Conditions of Sale, which are available upon request.

**Specifications are subject to change without notice.**

---

**Vicor Corporation**

25 Frontage Road / Andover, MA 01810  
Tel. 978.470.2900 / Fax 978.475.6715 / [www.vicorpower.com](http://www.vicorpower.com)  
Applications Engineer 800.927.9474

