Building a Wireless Power Transmitter

Illya Kovarik   *WPTX*

ABSTRACT

Wireless Power is rapidly gaining momentum in the market place enabling a convenient new way to charge mobile devices. Qi compliant Wireless Power Transmitters (TX) present the foundation on which wireless power infrastructure is currently being built. This report aims at helping wireless power developers to make their first steps building practical wireless power systems. It outlines a step by step approach, highlights dos and don'ts, and provides reference material on system troubleshooting and operating waveforms.

For more information on Texas Instruments Wireless Power product portfolio, including links to FAQ, please visit: www.ti.com/wirelesspower

CONTENTS

Introduction:

Building a Wireless Power Transmitter base for Qi compliance does not present any new or unusual challenges to the electrical engineer. This report outlines some practical steps to take both before and after the design with some troubleshooting tips in the event things do not go as planned. Comparison waveforms are included at the back of this report.

Design Capture:

The customer application schematic will typically closely follow the Texas Instruments recommended application design. It is, of course, of utmost importance that the application design be faithfully reproduced within the customer's CAD design workflow. The customer is encouraged to start directly from the Texas Instruments bq500xx Product Evaluation Module (EVM). The actual project design files can be provided in PADs or imported as an ASCii file.

Deviations from the recommendations, while possible, should be weighed and the potential risks carefully considered. The design capture should pass automated design rule checks without errors. It is also highly recommended to hold peer reviews at every significant stage of the design process.

Texas Instruments also offers to review customer's application schematic. This is intended to help guide customers through the process and to screen for obvious errors. The TI sales and application team in your area is there to help.

A first prototype should include basic test points on key signal nodes to aid in initial debug. A minimal surface mount (SMD) pad is adequate. Silkscreen identifying test points makes debugging and validation much easier and can be later deleted from production runs. At the very least the following test points are recommended:

- COMM+
- I_Sense
- PWM
- PM_Data, PM_Clk
- Vin, Vcc, 3.3Vcc, Vgate
- GND post (Multiple scope channels might be used at once)

Help with debugging is greatly facilitated when screen captures from key signal nodes can be shared on the Knowledge Base. Your screen captures can also be used to refer and compare to the screen captures at the back of this report.

Certain components and schematic details are critical to success and deserve special mention:

- The resonant tank capacitors
    - o Value must be correct or system will not regulate. There is no option.
    - o Dielectric must be C0G/NP0 type or efficiency will be low and COMM might fail. This must be followed.
    - o Voltage rating recommendations should be observed. Tank voltages can swing very high and even higher still with slight misalignment. Unfortunately, in the required

C0G/NP0 dielectric, there are also fewer voltage increments and the next jump up can appear too much. Required voltage ratings also depend on the product construction and how well consistent alignment can be achieved. Product reliability expectations also vary depending on the end application. Taking all these factors into account, it can be possible to down-rate the voltage rating of ceramic capacitors.

- o **Please follow any additional recommendations in the product datasheet!**
- The MSP430G2xxx Low Power Supervisor should be used if the final system requires a standby power of <90mW. Without it, a standby of approx. 300mW can be expected.
- MSP430G2xxx Low Power Supervisor RST cap value 1nF? The optional MSP430 can not be programmed by the user. It is automatically boot-loaded by the bq500xx at first power up. The wrong value of capacitor will low-pass filter the serial data stream from the bq500xx and possibly corrupt the upload process.
- /RESERVED pulled to GND with 10kΩ?
  - o This is needed to avoid false resets
- Sense resistor value correct?
  - o This should be 20mΩ 1% +/- 200PPM on all bq500xx systems.
- Current sense amp gain correct?
  - o This should be 100x on 19V system input voltage
  - o This should be 50x on all other system input voltages
- Other current sense resistor/amplifier combinations are useable but efficiency will be lower or noise higher. The components used on the Texas Instruments EVM have proven to be a good compromise and are readily available through distribution channels.
- If alternate MOSFETs are to be substituted:
  - o Recommendation for 19V System: 30V/4A FET with Rds-on < 40mΩ
  - o Recommendation for 12V System: 25V/4A FET with Rds-on < 25mΩ
  - o Recommendation for 5V System: 12V/6A FET with Rds-on < 10mΩ
  - o Depending on FET gate capacitance, the gate resistor might need to be adjusted
    - ▪ Higher value of gate resistor will improve EMC but decrease efficiency.
    - ▪ Wrong values of gate resistors high- or low- side can cause shoot-through.

T_Sense must be held above 1V for normal operation. This input is labeled 'T_Sense' but in actuality, it can be any kind of shut-down mechanism, crossing below the 1V threshold causes shut down. Thus, an negative temperature coefficient (NTC) sense resistor for thermal shutdown is but one of any number of possibilities. Standard applications do not require the added NTC sensor and added cost, the bq500xx incorporates current limit and self protection features. If NTC shut-down feature is not used, this pin must be pulled to the 3.3V supply of the bq500xx via a 10k resistor.

PCB Layout Tips:

Proper printed circuit board (PCB) layout is critical to the success of the application. A poor layout can cause the whole application to not work properly. Beyond basic circuit operation, the PCB layout also directly influences the Electro-Magnetic Compatibility (EMC) profile. Therefore, greatest care should be exercised during the PCB layout stage of the project.

All Texas Instruments TX EVM's pass EMC CISPR22 and FCC PART 18 requirements because basic PCB design rules were followed. Wireless Power TX bases do not have any unusual or specific

requirements but systems do vary depending on the application. There are many references on proper PCB layout techniques with regards to power supplies.

Again, it is highly recommended to hold peer review at every major stage of the design process, especially the PCB layout. The PCB design review is a sort of comprehensive design review, since the next step is commitment to hardware. Any changes thereafter mean rework which can be painful.

A few good PCB design tips are repeated here with references to various revisions of Texas Instruments system evaluation hardware used to validate the product:

Tight loops!

For higher system input voltages, a DC-DC buck regulator will be used to step down the 12 or 19 V input to the 3.3 V supply to the bq500xx. A linear regulator could be used but most applications can not afford the added dissipation, which could be more than ¾ Watt, hence the buck regulator. With such a step-down ratio, switching duty-cycle will be low and the regulator will be mostly freewheeling. Therefore, place the freewheeling diode current loop as close to the switching regulator as possible (loop in red). Place the buck inductor and power loop as close to that as possible (loop in blue):
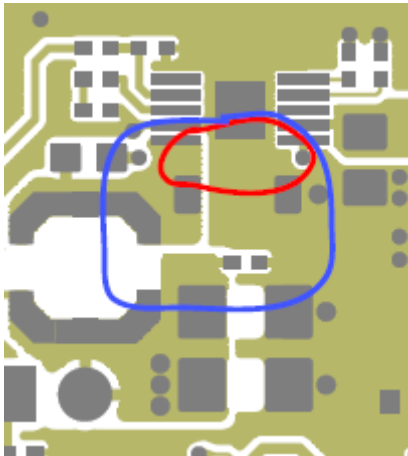
The same applies to the half bridge switching FETs:
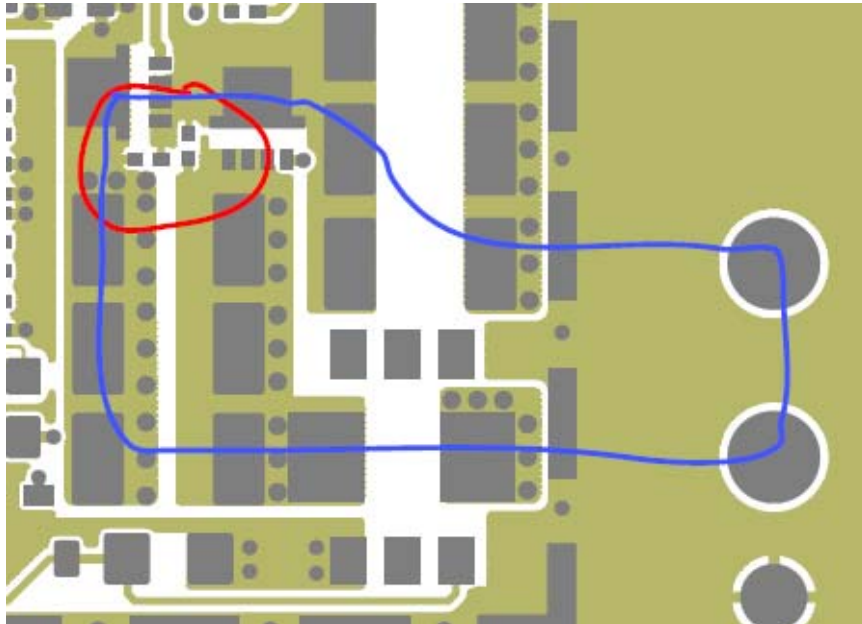
Ground Planes!

Generally speaking, the TX layout will require a 4 layer PCB.

One proven approach to the layer stack-up, as used in the Texas Instruments EVM, is as follows:

Layer
1   Component placement and as much ground plane as possible
2   Clean uninterrupted ground copper pour
3   Finish routing
4   Clean ground, or minimal finish routing, as absolutely needed

Thus, the concept is to virtually sandwich the circuitry between grounds. This provides a perfect ground reference plane and also helps to contain and minimize EMI (electro-magnetic interference) noise emissions.

A 2 layer PCB can be possible but will require advanced PCB layout experience and likely several board revisions to achieve satisfactory performance. The main problem with the 2 layer PCB lies in achieving the solid ground reference for the bq500xx. Failure to do so can result in anything from failing EMI testing to more subtle processor clock tolerance issues caused by the inadequate ground reference. Having a processor clock which is not accurate can lead to drive frequencies being different than expected, poor regulation, or failing of Qi Certification timing requirements. The cost savings might not be worth it.

Grounding of unused pins, if there is a choice, is good practice. Some have not been grounded on the EVM to allow for greater flexibility in the field. On the other hand, if a 2-layer board is attempted,

grounding the unused pins and continuing the ground flood around the bq500xx could be critical to success.

In addition to this layer stacking technique, one can place an SMD mounted metal box (e.g. Laird P/N BMI-S-207-C) to further shield the components. In practice this shield is not needed and the EVM performs well in typical regulatory emission tests. Note that the perimeter ground via stitching further encapsulates the circuitry:


Bypass Capacitors on the bq500xx!

Make sure the bypass capacitors intended for the bq500xx Vcc supply are actually bypassing these supply pins (pin 33 & pin 36) to solid ground plane. This means they need to be placed as close in to the device and traces widened as much as possible. An excellent example is shown here:
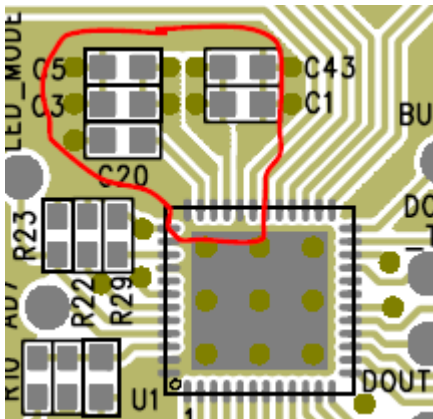
Keep as much copper as possible!

The PCB traces start fabrication as a sheet of copper which is then etched to leave the desired circuit patterns. Thus, there is no added cost in leaving more copper; use it to best advantage! Make sure the bq500xx has a continuous flood connection to the ground plane (see red arrows). Also, it is good practice to stitch the PowerPad under the bq500xx with vias to an inner ground plane:
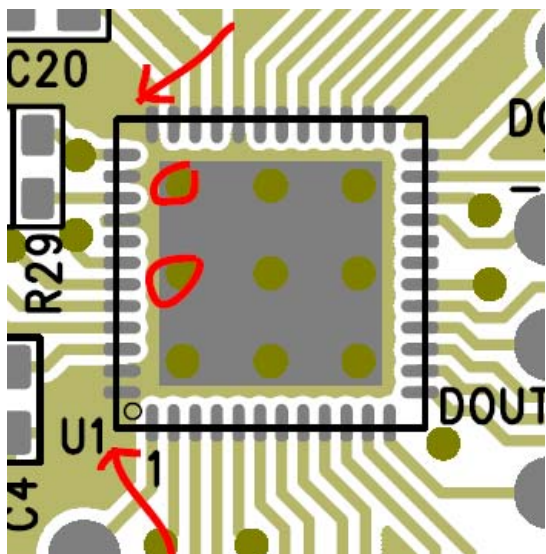
The copper ground plane also provides minimal heat sinking. In practice, the bq500xx never does get warm. In other words, something is wrong if it does get warm.

Separate the analog control ground plane and each power section ground plane and use only ONE tie point to tie them together. Having several tie points between the same sections defeats the purpose of separating the grounds. NOT separating the ground planes can lead to analog sense signals bouncing on power return and overall erratic system performance:
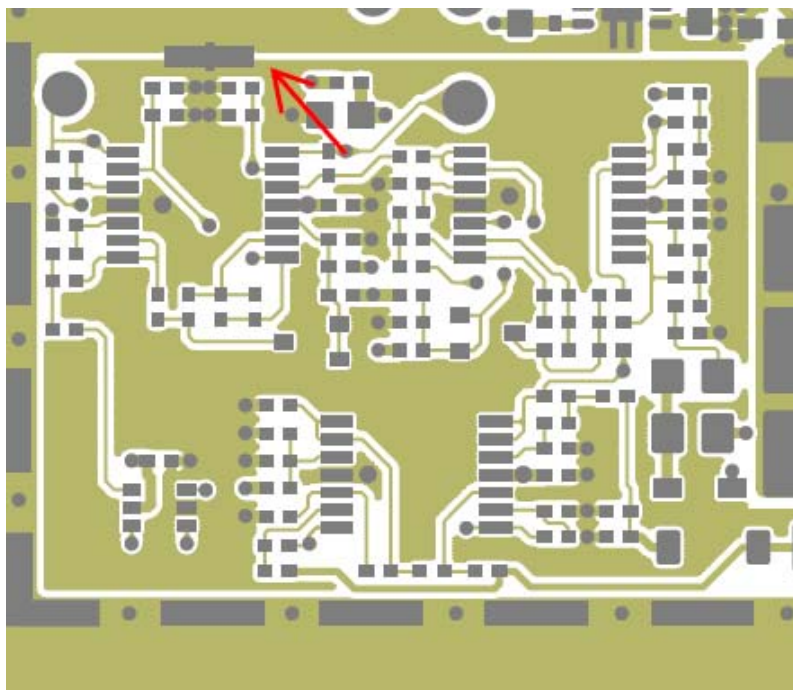
As demonstrated on the web orderable Texas Instruments EVM, it is possible to place all components on one side of the PCB.

Taking this to the logical end point, the user application could "fold" the EVM over and place the coil on the top and all the components on the bottom. The result would be the smallest footprint possible for a Qi-compliant wireless base, since the coil size is mandated by the WPC Specification.

Proper Current Sensing Layout Technique!

When sampling the very low voltages generated across a current sense resistor, be sure to use the so called, "Four-wire" or "Kelvin-connection" technique. This is important to avoid introducing false voltage drops from adjacent pads and copper power routes. It is common power supply layout technique.

In the below screenshot of a Texas Instruments PCB layout, the current sense resistor is R21 and input current is flowing from left to right. Notice how the R18 and R22 sensing resistors are connected to the pads of R21 so there is no measurement error introduced by copper conduction losses or copper resistance temperature dependency.

Proper and accurate current sensing technique is critical to the correct performance of parasitic metal loss detection algorithms such as PMOD (parasitic metal object detection) and FOD (foreign object detection). The sense resistor component selected should also be appropriate for the task. In particular, it should be 1% tolerance and have a temperature stability of at least 200PPM.

Building a Wireless Power Transmitter
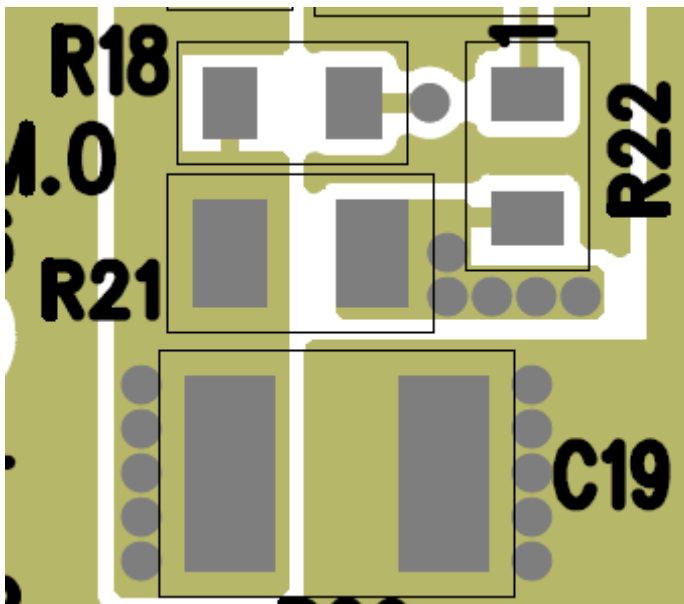
Balanced COMM line!

COMM+/COMM- sense lines should be run as a balanced or differential pair. The WPC packet information runs at 2kBaud which is essentially audio frequency content and this balancing reduces noise pickup from the surrounding switching power electronics. There is no need to tune or impedance-match these lines as would be the case in RF signaling.
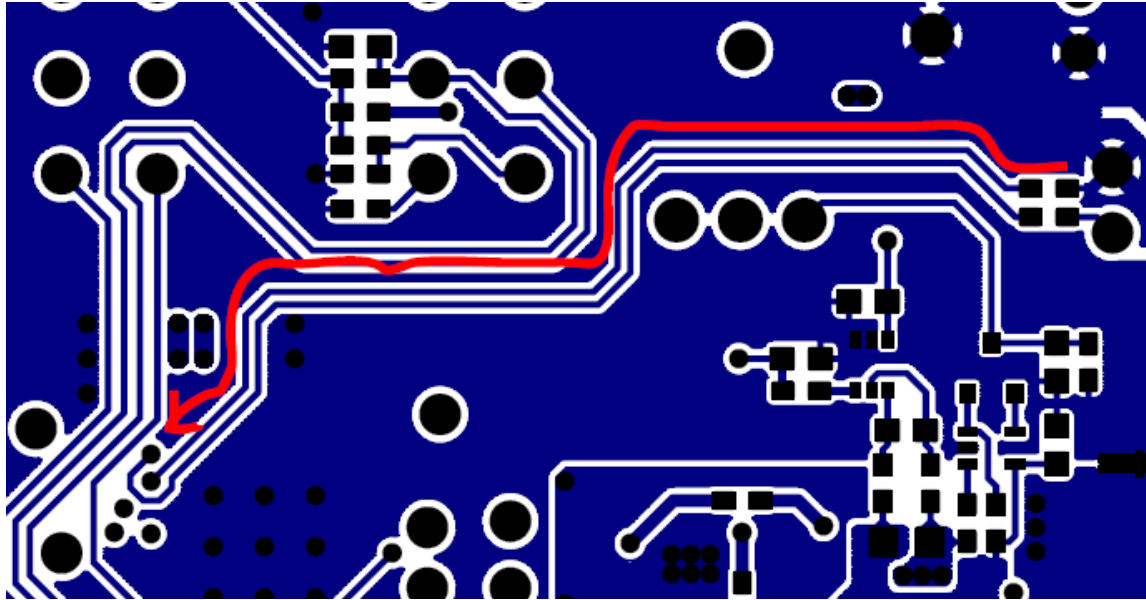
Tuning Component Values on the Prototype!

Some tuning must be done on real hardware, while testing for lowest emissions or highest efficiency, and there are the following "knobs" to turn:

FET gate drive voltage
FET gate resistor
Capacitor snubber across FET

There are no hard-and-fast values to be computed or simulated. Some component value adjustments will almost always be required in circuit while testing. These adjustments are subtle tradeoffs as lower emissions can come at the expense of lower efficiency, so caution is advised. Some general guidelines can be provided:

Keep the gate drive regulator voltage to a minimum practical value. Using the lowest cost Texas Instruments MOSFETs suited to the 19V low power application, the NexFET CSD17313Q2, we consult the datasheet concerning the gate voltage vs. RdsON graph:
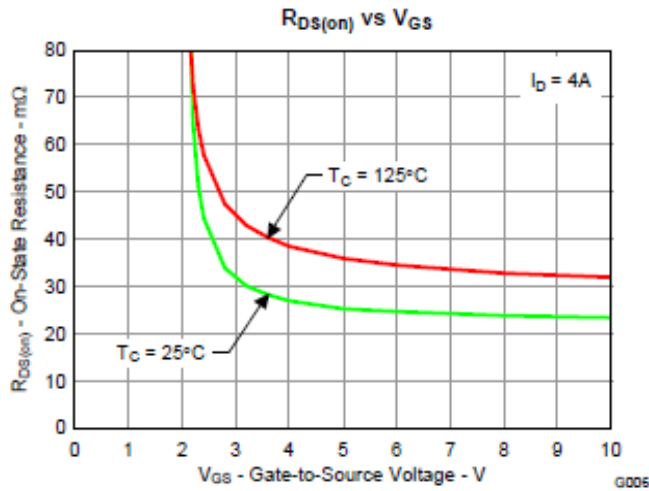
As the above graph shows, driving the NexFET gates with a voltage greater than 5 to 6 Volts does not substantially improve on-state resistance.

The Texas Instruments EVM uses the most primitive 'regulator' to derive the gate drive voltage from the input supply. It matches the bare essentials of the requirement; smooth filtered voltages are not required to enhance a MOSFET gate. The user can certainly use a dedicated voltage regulator, especially if there is other auxiliary circuitry to be powered.
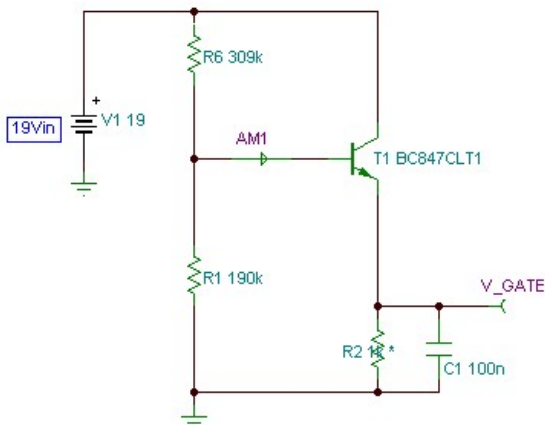
The MOSFET gate drive resistors might need to be adjusted. In particular, the upper FET drive resistor (R56 in below figure) will serve to slow down the NexFET fast switching action, thus reducing noise. It is important to note that since the gate capacitance of the NexFET devices is so much lower than industry standard parts, a much larger value of gate resistor will be needed. This particular early Texas Instruments System Board was assembled with a 25 Ω gate drive resistor (R56) and did very well in EMC pre-compliance screening.
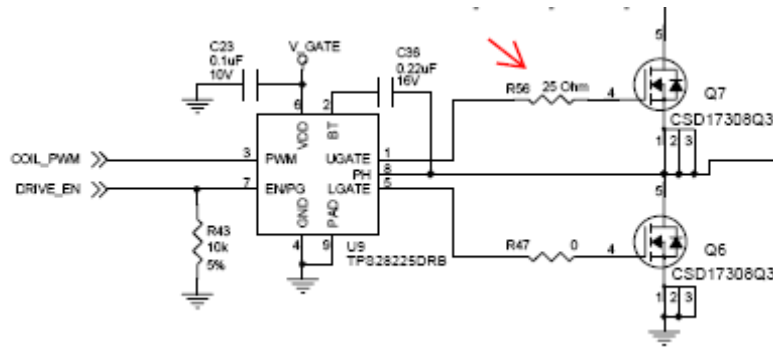
The gate drive resistors themselves also serve as test points to observe the actual drive waveforms. Do not add any extra test points to the line from the MOSFET driver to the transistor gates; possible parasitic inductance or stray capacitance is unwelcome. Never add a current probe to the gate drive line! Do not add test points or resistance to the bootstrap capacitor either. Lower the gate drive supply voltage rather than adding resistance to the bootstrap circuit. In other words, keep it short, simple and direct.

If for some reason MOSFETs other than the recommended Texas Instruments NexFETs are used, pay careful attention to their correct application. Critical MOSFET criteria are repeated here:

- If alternate MOSFETs are substituted:
   - Recommendation for 19V System: 30V/4A FET with Rds-on < 40mΩ
   - Recommendation for 12V System: 25V/4A FET with Rds-on < 25mΩ
   - Recommendation for 5V System: 12V/6A FET with Rds-on < 10mΩ

Gate capacitance varies by MOSFET device chosen. The gate resistance and the gate capacitance form an RC time constant which influences the on/off switching times. Using MOSFETs with a substantially higher gate capacitance, yet retaining the gate resistance recommended for the Texas Instruments NexFETs, can lead to shoot-through. Verify by checking the high-side and low-side PWM drive waveforms then adjust the gate resistance, typically by reducing their value, bearing in mind that the alternate MOSFETs will usually have a higher gate capacitance.

Turn-off diodes anti-parallel to the gate resistors are unnecessary at these low power levels. The simple resistor, properly chosen, is adequate.

An item worth mentioning is the capacitor snubber on the main switching FETs:
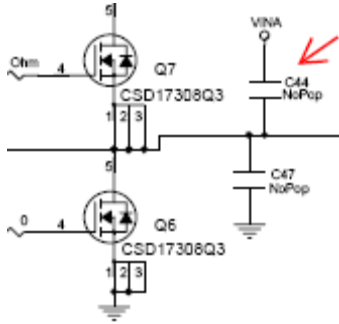
Such capacitor snubbers must be placed as close to the switching FET as possible, otherwise parasitic PCB trace inductance will defeat the purpose:
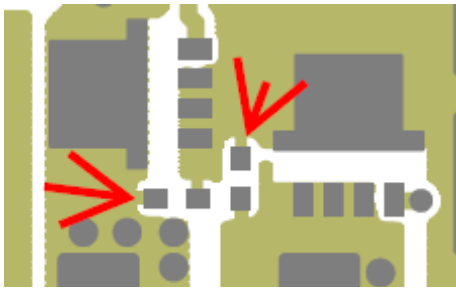
Earlier generation Texas Instruments TX EVMs included this capacitor snubber. It was evaluated but has since been generally omitted from all application schematics. Customer systems may vary however, and one might want to consider this valuable tuning aid on the PCB layout to reduce EMC emissions. It can always be left off the Bill of Materials if not needed.

Powering Up and Testing New Hardware:

Once the new boards have been assembled and before applying power, the boards should undergo a thorough visual inspection with a microscope.

The 19V bq500xx application requires at least a 19VDC/525mA (10 Watt) input supply for proper operation. At first power up, it can be advantageous to limit the input current to a lower level should there be an unintended short circuit on the board. Follow normal power supply bring-up procedure.

The best starting point to test a new TX assembly is with a known-good cross-reference such as a working Texas Instruments RX EVM (bq51013 HPA725 RevA). Apply power to the TX and place the RX EVM on the new TX charger pad, COMM and power transfer activity should start instantly and the RX should come to life without an exterior load first. Once basic COMM has been reliably established, an external load can then be incrementally added to the RX EVM up to the full 5 watt rating. Make sure the supply is not current limited from the source when testing full load.

Anticipating frequent testing of new TX assemblies, it can be convenient to build a test RX with a common 2 watt load resistor permanently attached. A value of 50-100 Ω is a good choice, providing a minimal load, and making a handy gage for quick system verification.

Dedicated WPC TX protocol test equipment has recently become available for purchase. The Texas Instruments bq500xx product line has been designed to be user friendly and standard bench instruments are sufficient for design validation testing: Supply, Load, Multimeter and Oscilloscope. Following the recommended application and the tips in this report, developers can be confident in submitting their product for Qi certification.

Some comments with regards to electronic loads used on the test bench, the default input mode is usually *constant current*. This can sometimes cause startup problems, especially into a fully loaded condition. This will depend on the type of electronic load, lead length, and can be worsened by voltage drops within the application hardware. Obviously, it is easy to reduce the load or to switch to *constant resistance* mode and retest. Should the startup issue go away, better on-board supply voltage bypassing might be needed.

Once running, further measurements can be made. Example voltage node waveforms are included at the end of this report for comparison.

CAUTION:
Do not clip oscilloscope grounds while ANY wireless power system is energized. Always clip on the ground *before* power is applied! The TX and RX are *isolated* and can be at very different potentials and the sudden conduction path formed by the ground clips can damage components. With the common ground established *before* power is applied, probing the different circuit nodes can proceed normally.

Construction Details:

Verify the power adapter is adequate for the task. For a 19V low power system, the adapter needs to be capable of delivering at least 19V 525mA (10 Watt) to the TX. The voltage drop in DC feed line should not cause the TX input voltage to drop below the 18V WPC minimum input voltage specification for the A1 type transmitter.

Housing spacing between coils is critical. According to WPC specification, there must be a 5mm maximum spacing between the TX and RX coils. The end-product plastic housing, retaining pockets, etc., must be designed accordingly. The minimum modulation test often fails when this spacing is not held.

The TX and RX coils need to be aligned for proper operation and to obtain best efficiency. Even in a system with the magnet attractor, some slight misalignment can be tolerated and the system will still regulate. The degree of misalignment with magnet attractor, however, is not quantified in the WPC Specification.

Metal Product Housings and PMOD/FOD False Trips:

PMOD and FOD are safety features designed to monitor the "live" system efficiency by comparing power transferred with power delivered. When the efficiency difference exceeds a threshold, the TX stops transferring power.

Thus, be aware that PMOD/FOD mode might be protecting against "friendly metal" in the intended housing construction. The bq500xx can not tell where the additional metal object loading came from.

Consider if there would be any reason for parasitic metal protection to trip in the first place or whether there is any metal in the housing construction. Verify there are no metallic assembly objects or hardware in the TX/RX coupling path.

PCB ground planes can become parasitic metal if positioned too close to coil coupling path. A round metal cut-out that just clears the coil edges can actually become a shorted turn of a parasitic secondary if the clearance hole is too small.

NOTE:
All PMOD/FOD performance suspicions can be verified by simply removing the PMOD programming resistor and retesting. Removing the programming resistor disables the PMOD feature.

On the other hand, metal and metalized shielding can be used very effectively to improve system performance. Putting a ground plane under the ferrite shield will actually improve EMC performance by absorbing the reverse radiated field.

The EVM uses an aluminum plate under the coil. This forms a ground plane behind the coil to reduce EMC. The aluminum plate itself is an inexpensive but conductive spacer which enables catalog standoffs to capture the coil and ferrite between the PCB bottom and the clear plastic top. Using the ground plane and aluminum plate behind the coil is another case where metal can improve

performance since it catches stray EMC emissions. The EVM passes regulatory standards for EMC CISPR22 and FCC PART 18.

Multiple TX Circuits per Charger Base Unit:

Be aware of the possibility of cross-talk between adjacent units if 2 or more TX circuits are combined into one common dual bay charger unit. Although quite possible to implement successfully, it needs to be validated on a per case basis, as product packaging and styling varies greatly.

Regarding Coils:

Please refer to: "bqTesla Transmitter Coil Vendors," SLUA649, for the most up-to-date list.

Please also follow any recommendations in the datasheet as each product has been designed to operate with a specific coil type, as set forth in the WPC specification. The specific controller/coil/capacitor combination must be used to be WPC compliant and achieve Qi-Certification.

Six of the Most Common Pitfalls to Qi Certification:

Following all the previously discussed recommendations will lead to a functioning TX base which can be confidently submitted for Qi-Certification and EMC agency approval. If, however, the unit fails in some particular test, below are some typical scenarios:

1) Plastic housing tolerances result in RX/TX coil spacing too large and the Qi minimum modulation test fails. (Must be <5mm)
2) Wrong resonant capacitor type and the Qi minimum modulation test fails (using X7R rather than C0G dielectric type to save cost can backfire!)
3) Insufficient ground plane and via stitching on bq500xx can cause timing or frequency requirements to be out of WPC spec (use 4 layer PCB and follow grounding recommendations!)
4) COMM signal not routed balanced and therefore noisy can lead to failing Qi guaranteed power test (compare COMM+ signal to example and probe the EVM)
5) Vcc bypassing on bq500xx, excessive ripple, can lead to failing EMC tests (compare to example waveform in the test and probe the EVM)
6) Metal housing parasitic losses, copper plane in field, can lead to PMOD or FOD failures (follow the recommendations)

Example Waveforms:
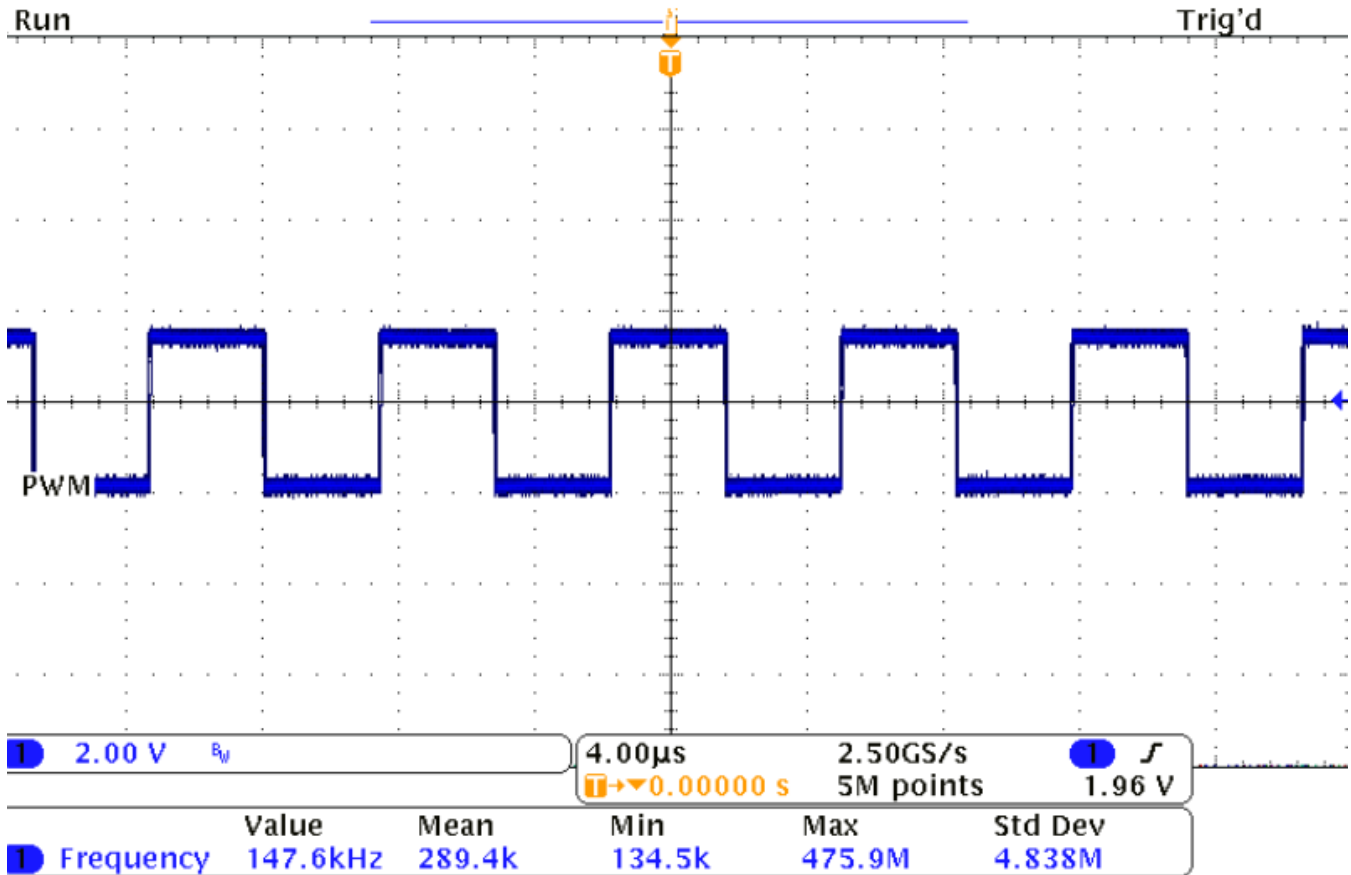
Typical PWM at 1/4 Load

A typical PWM drive signal as output from the bq500xx/QFN48 is shown. It will be 50% duty and should be in the range of 140kHz to 150kHz at the lighter load.
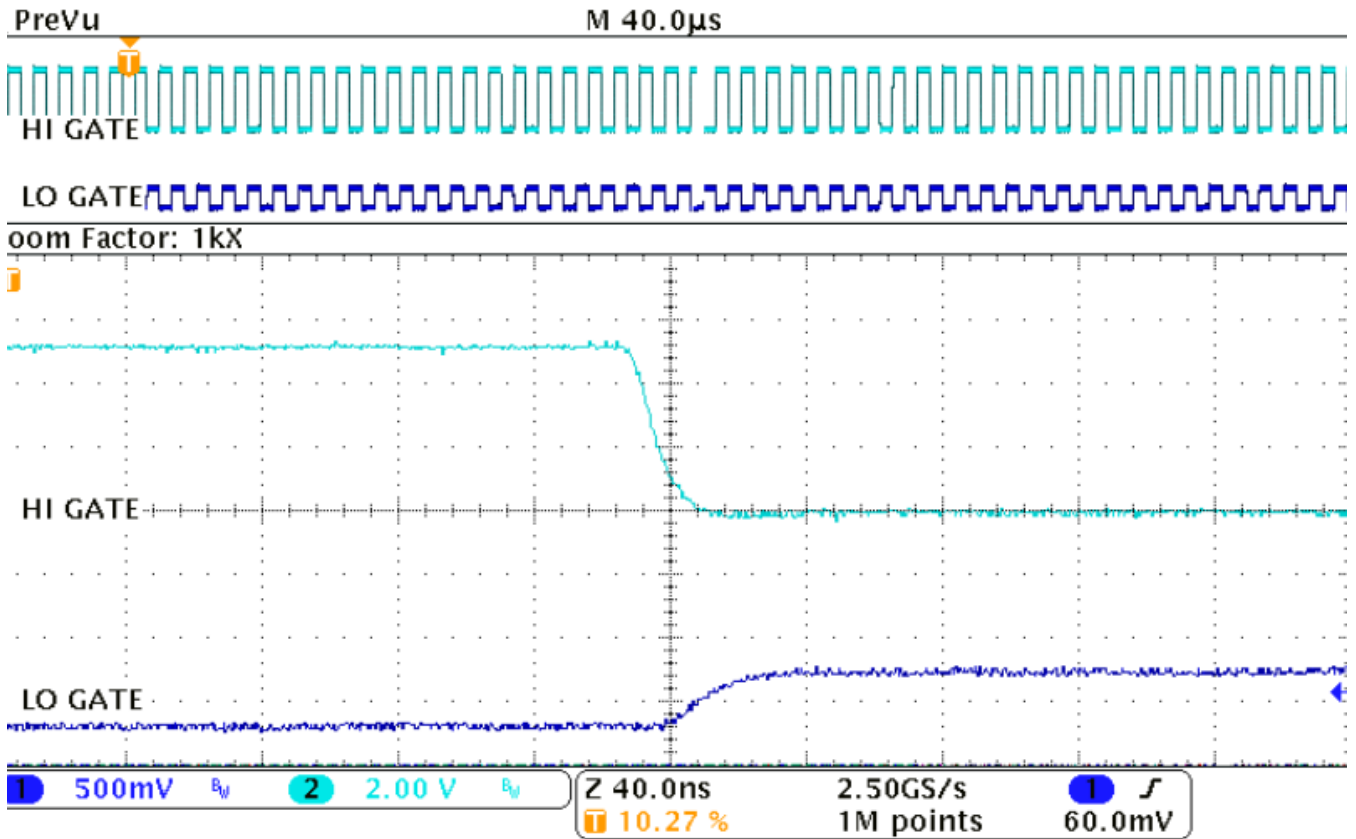
Gate Drive Waveforms

Look at the actual MOSFET gate drive signals to verify. Test points are not recommended on the gate drive lines, probe directly on the gate drive resistors. The upper MOSFET should be switching off before the lower MOSFET is switching on. The gate drive resistors are meant to slow the switching action and to reduce EMC, but taken too far, shoot-through will begin to occur and efficiency will be reduced. It is a balance!

Stop
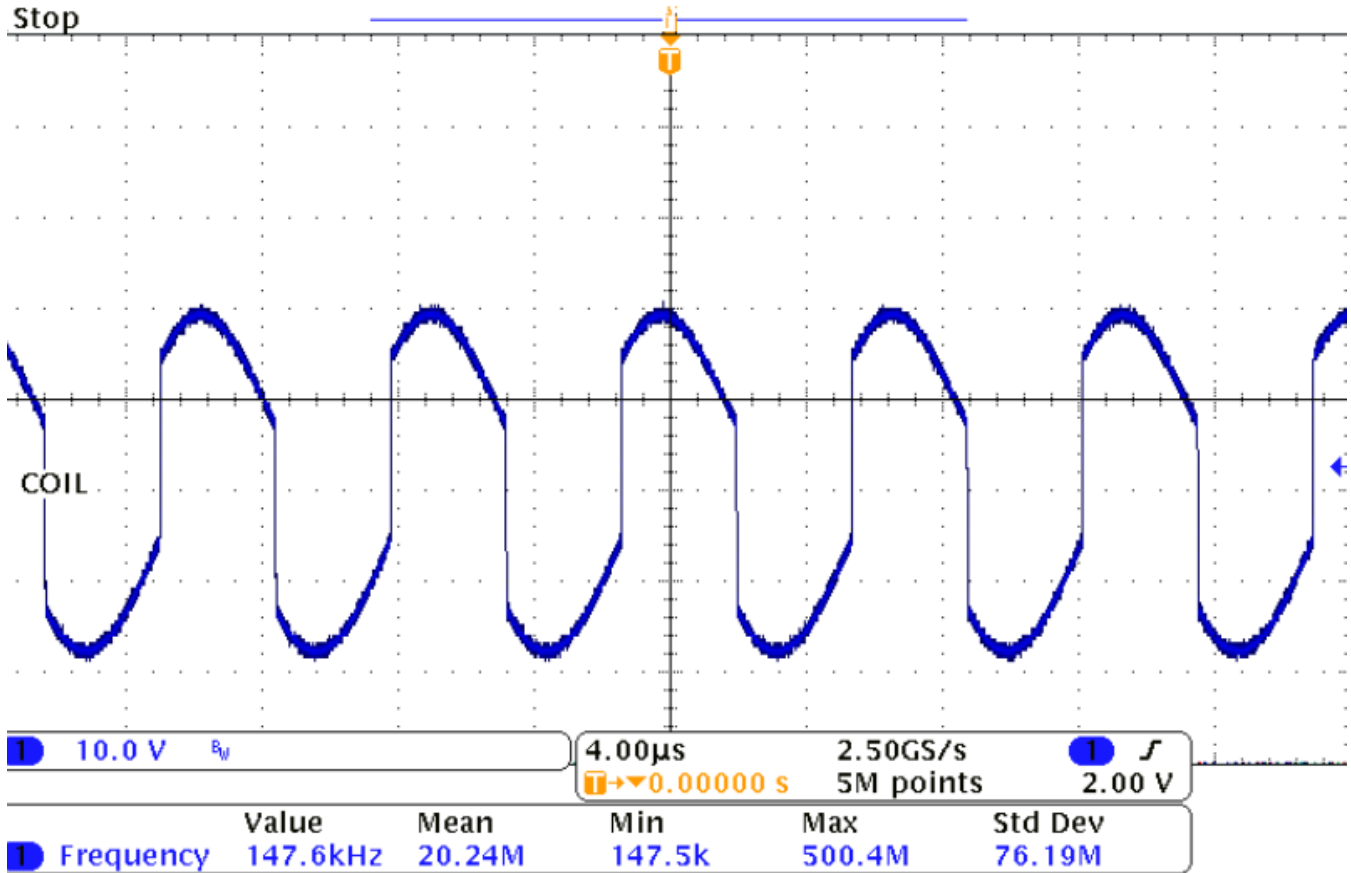
Typical Coil Voltage at 1/4 Load

The typical coil voltage is shown. The wave will not necessarily be sinusoidal, it depends on loading and the operation frequency. Note the high voltage swing.
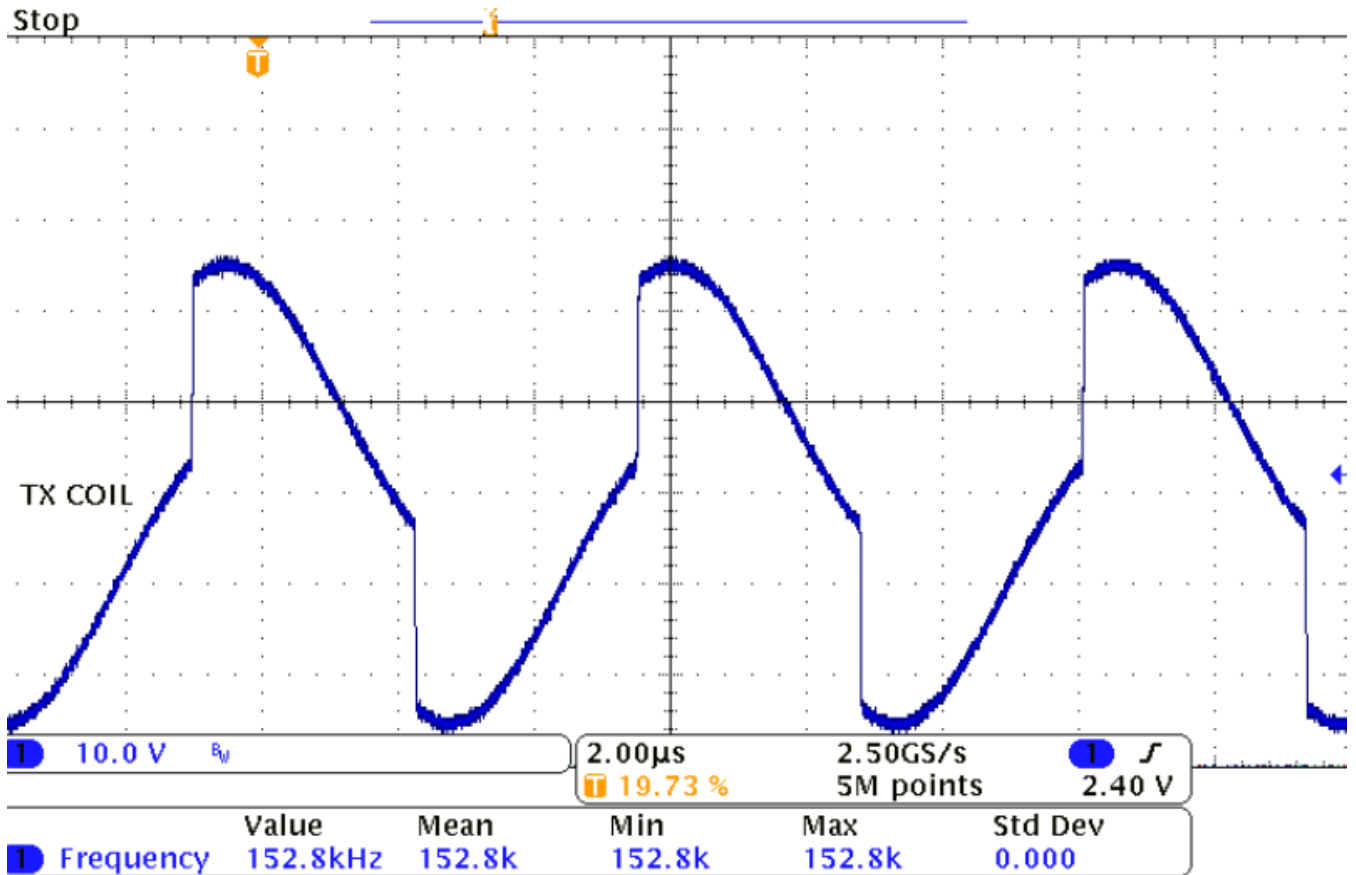
Typical Coil Voltage Full Load

The typical coil voltage is shown. Again, the wave will not necessarily be sinusoidal, it depends on loading and the operation frequency. Note the high voltage swing.
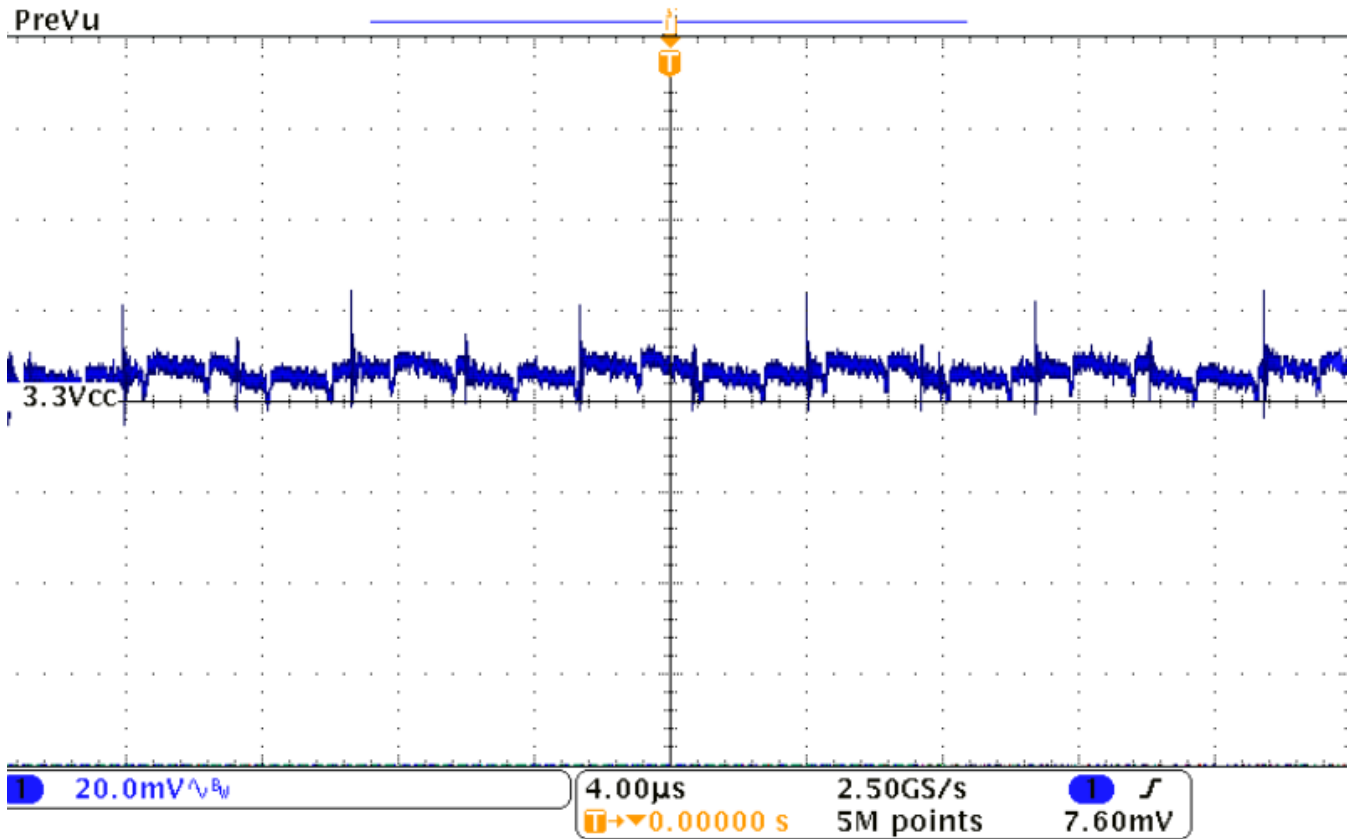
Ripple on 3.3Vcc Supply

It is necessary to supply the bq500xx/QFN48 with a clean 3.3V Vcc supply! Note, in this screen shot the scope is AC coupled, this would be the ripple on top of the 3.3VDC supply and it needs to be low. Make sure the bypass capacitors are truly bypassing the device.
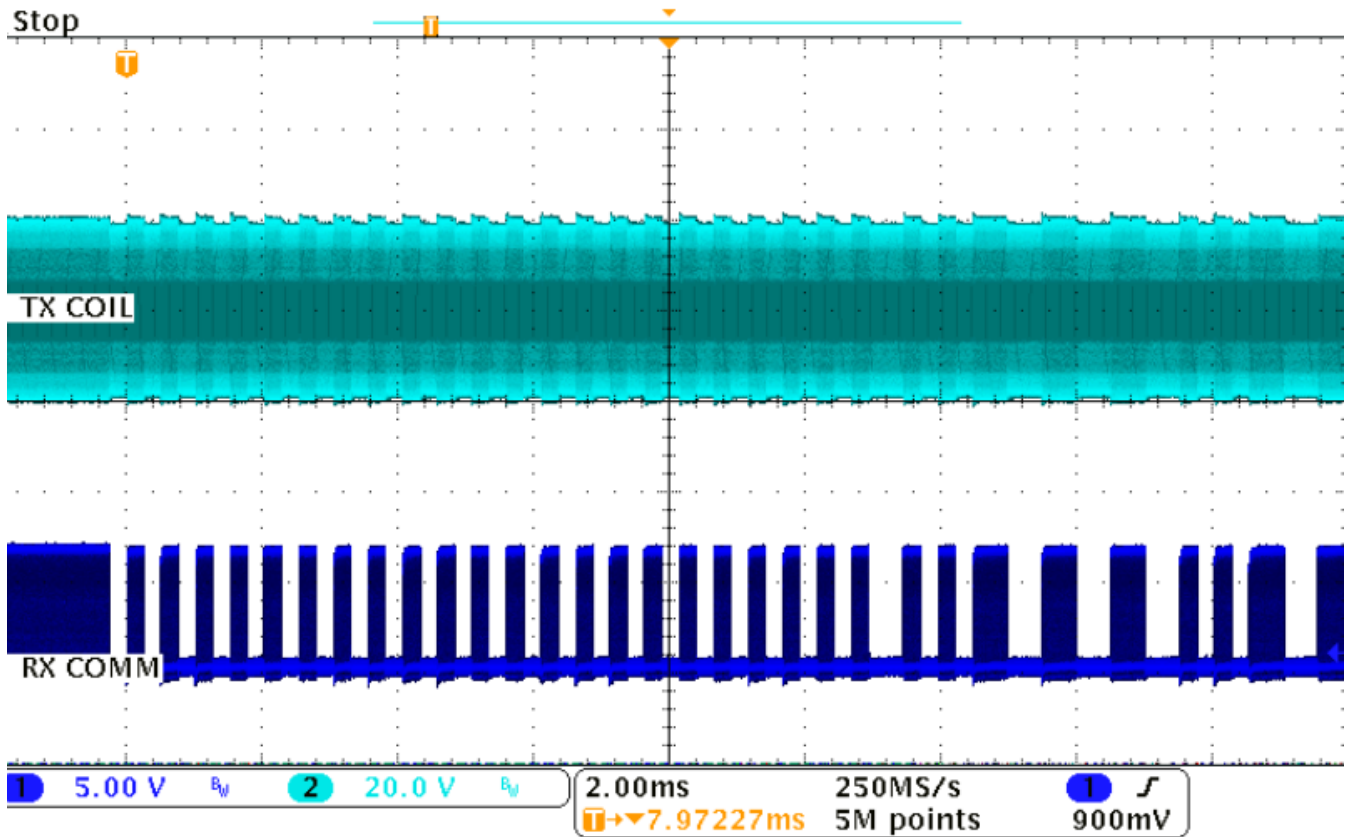
Typical COMM Signal

This is the typical back-scatter amplitude modulation from the RX as it is seen on the TX coil. This would be a typical WPC message packet from the RX, perhaps telling the TX to send more power.

This is triggered from the RX signal so that the TX amplitude modulation is easy to view. Remember when combining the TX & RX grounds to synchronize scope signals: **Make the ground connections before power is applied!**
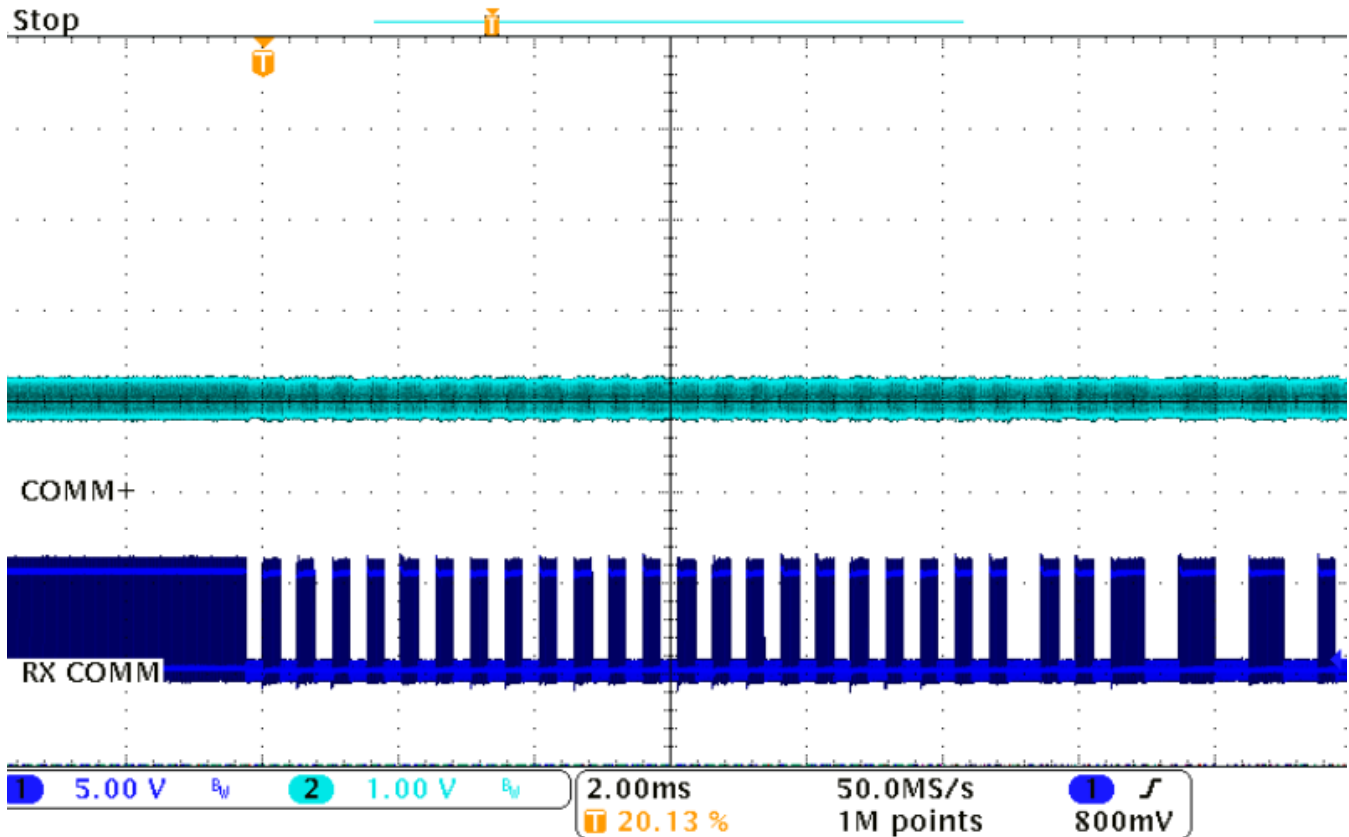
COMM+ must be centered 1V

The coil voltage signal is divided down to a more manageable level, and with a small filter and DC adjust it is the COMM+ signal read by the bq500xx/QFN48. The COMM+ signal must be centered around 1V and should never swing below ground. This is the communication link and good signal integrity is critical to success of the system. Please adhere to recommendations given in the text.
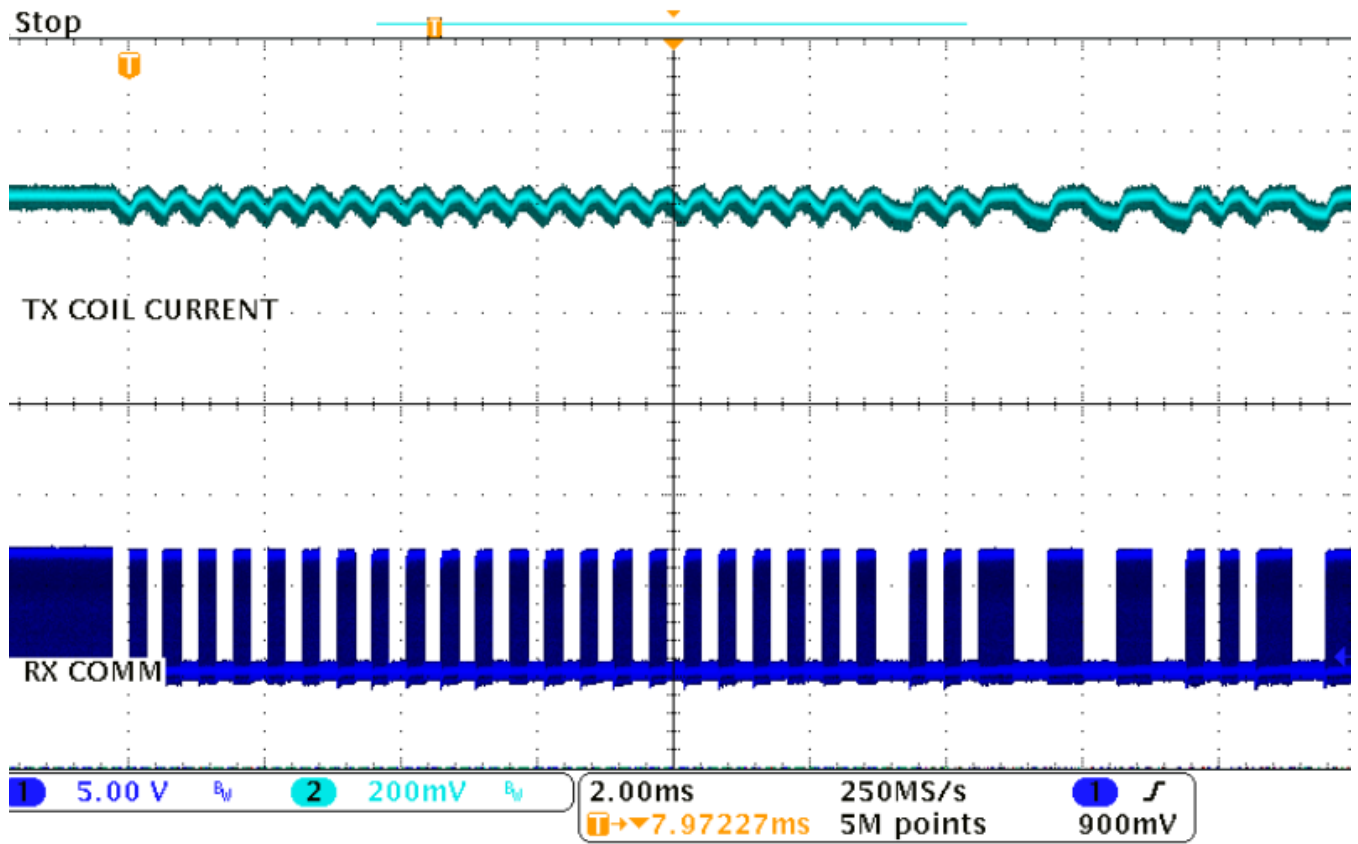
Typical I_SENSE Signal

The modulation can be seen on the I_Sense signal. The I_Sense signal should be a clean average without any abnormal spikes.
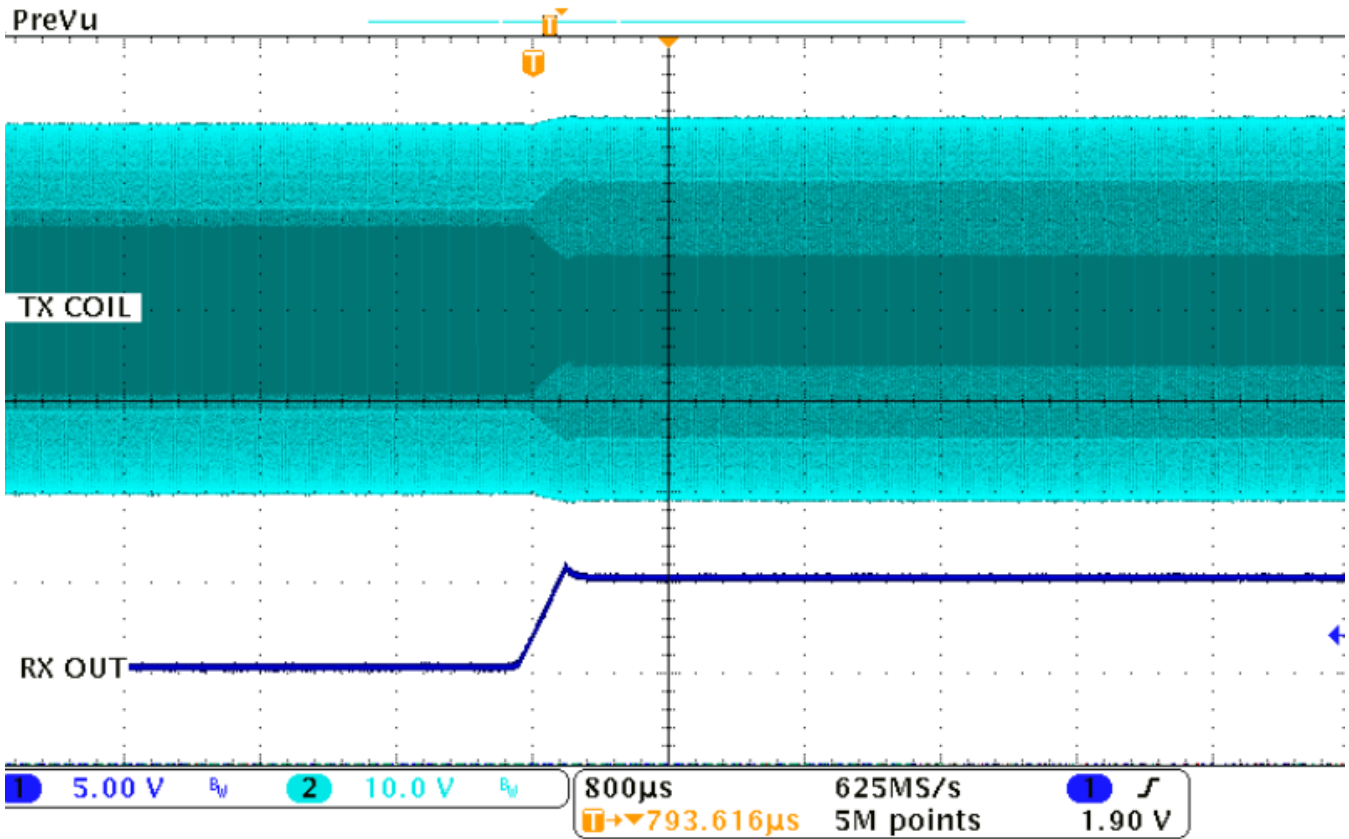
Typical RX Load Switched 'On'

The load on the RX side is switched on. The amplitude and frequency on the TX will change.
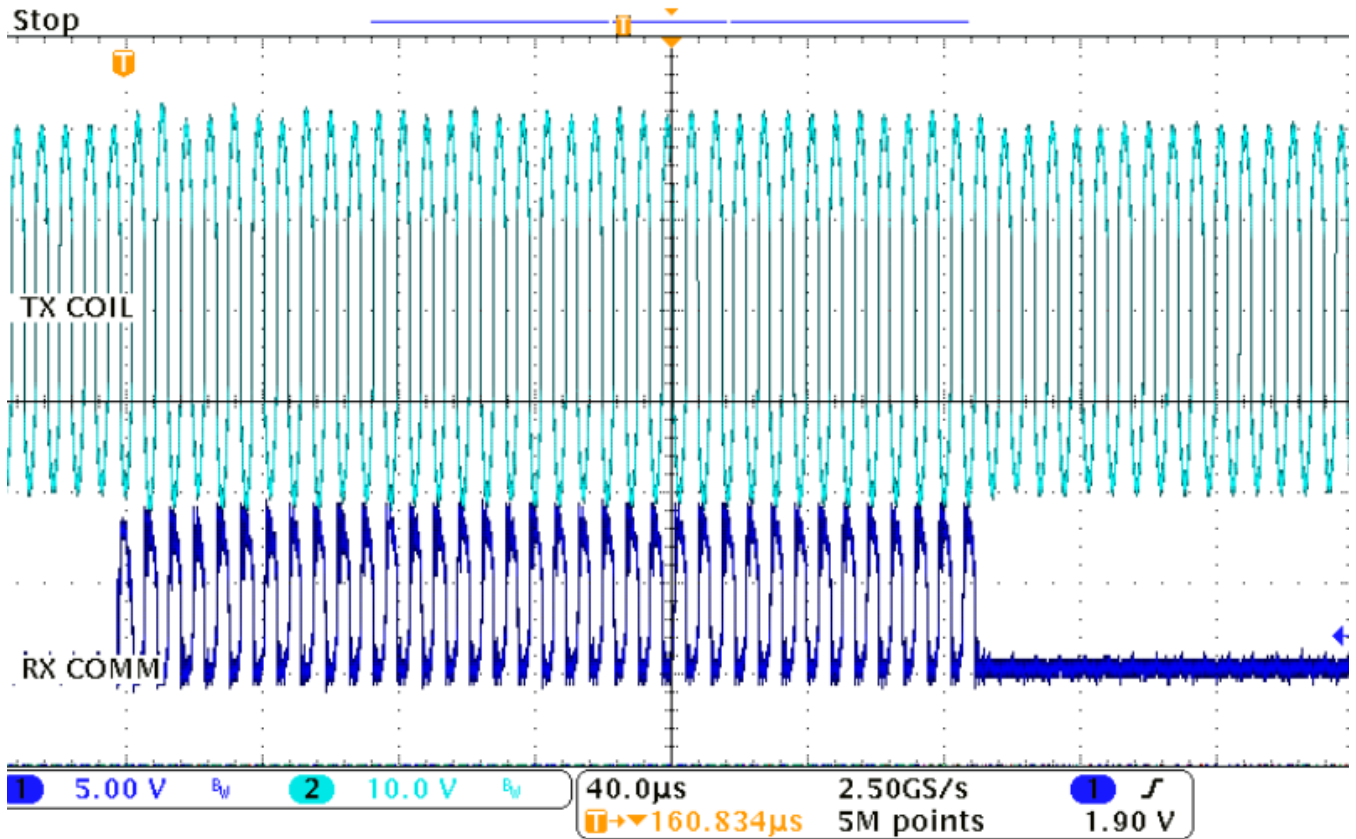
Typical COMM & Coil No Load

Zoom into the actual switching waveform is shown. The power switching waveform is the carrier of the WPC amplitude modulated waveform.
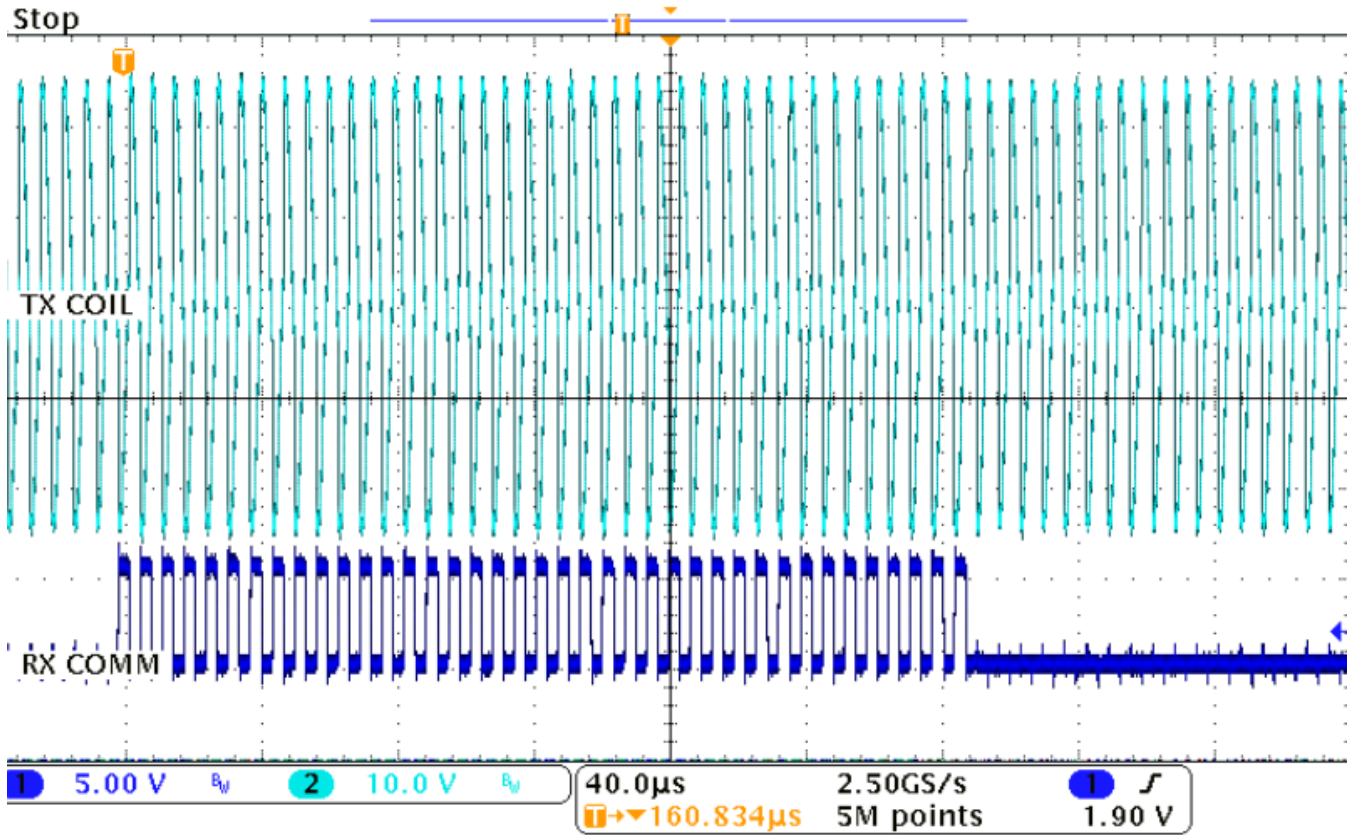
Typical COMM & Coil Full Load

Zoom into the actual switching waveform is shown but under full load. The power switching waveform is the carrier of the WPC amplitude modulated waveform.
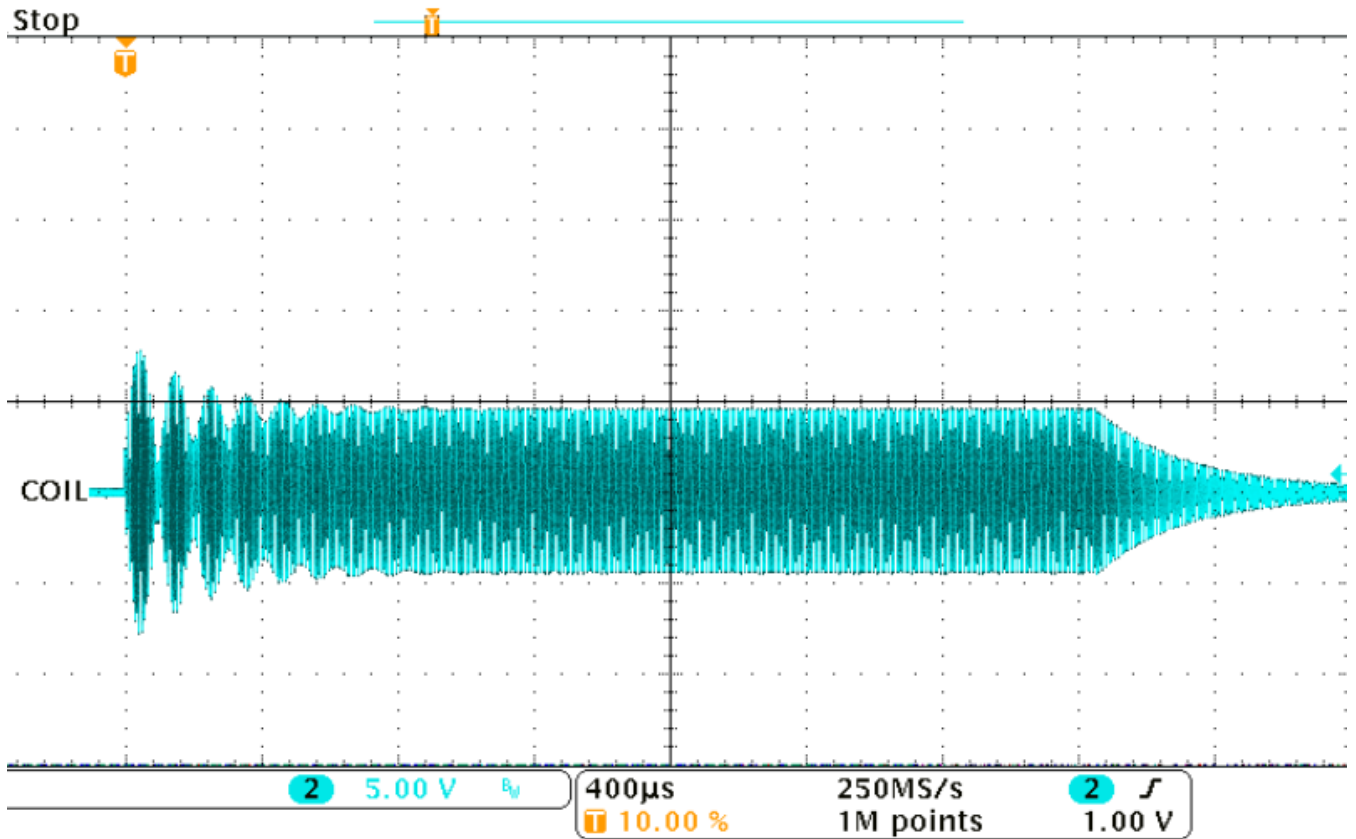
Analog Ping Cycle

This is the periodic analog ping cycle repeated every 400ms. The bq500xx/QFN48 monitors the amplitude of this wave and if an RX is present a change in resonance is detected. If such change is detected a further digital ping and COMM handshake takes place.
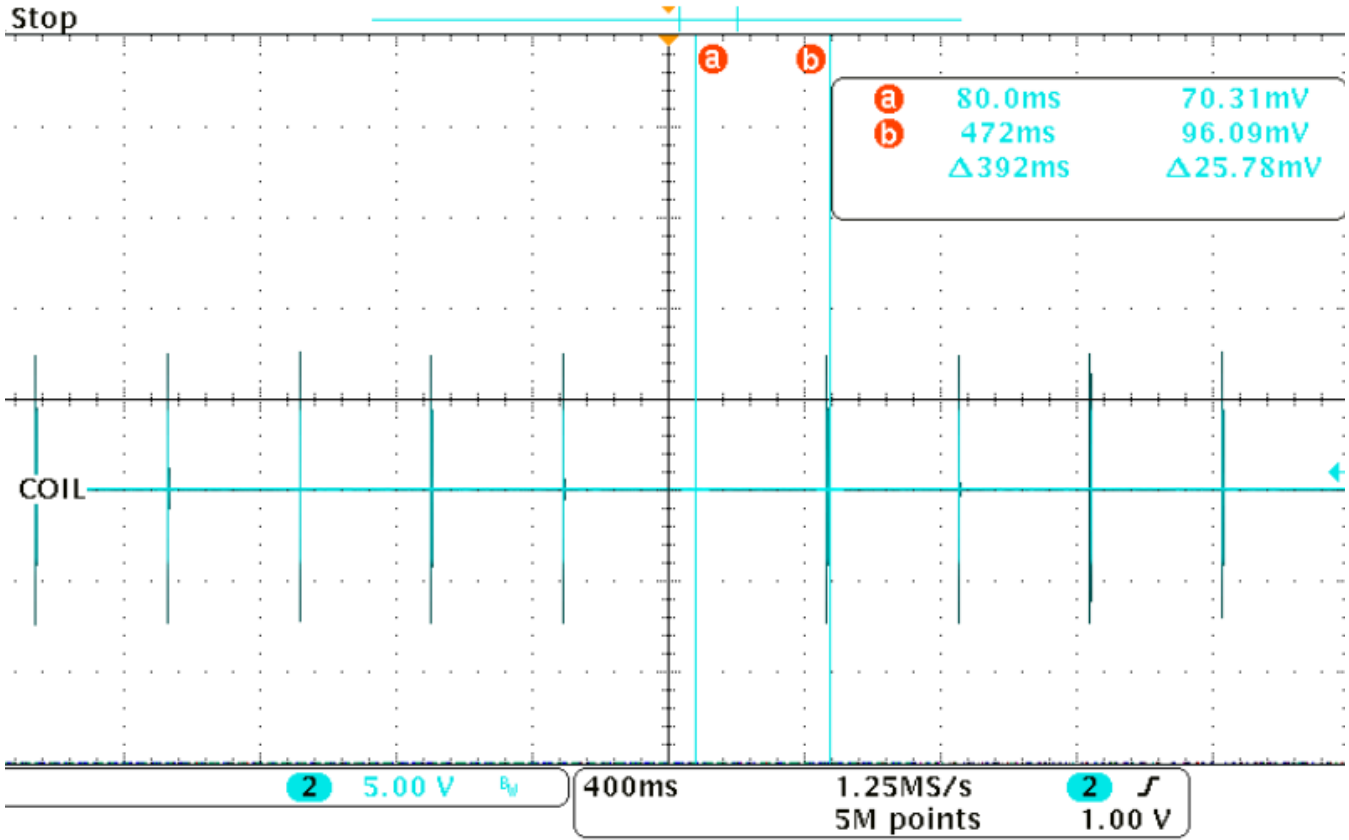
Analog Ping every 400ms

Panned out over time to show the ~ 400ms analog ping interval is shown. The ping interval and duty cycle are balanced to reduce quiescent power consumption.
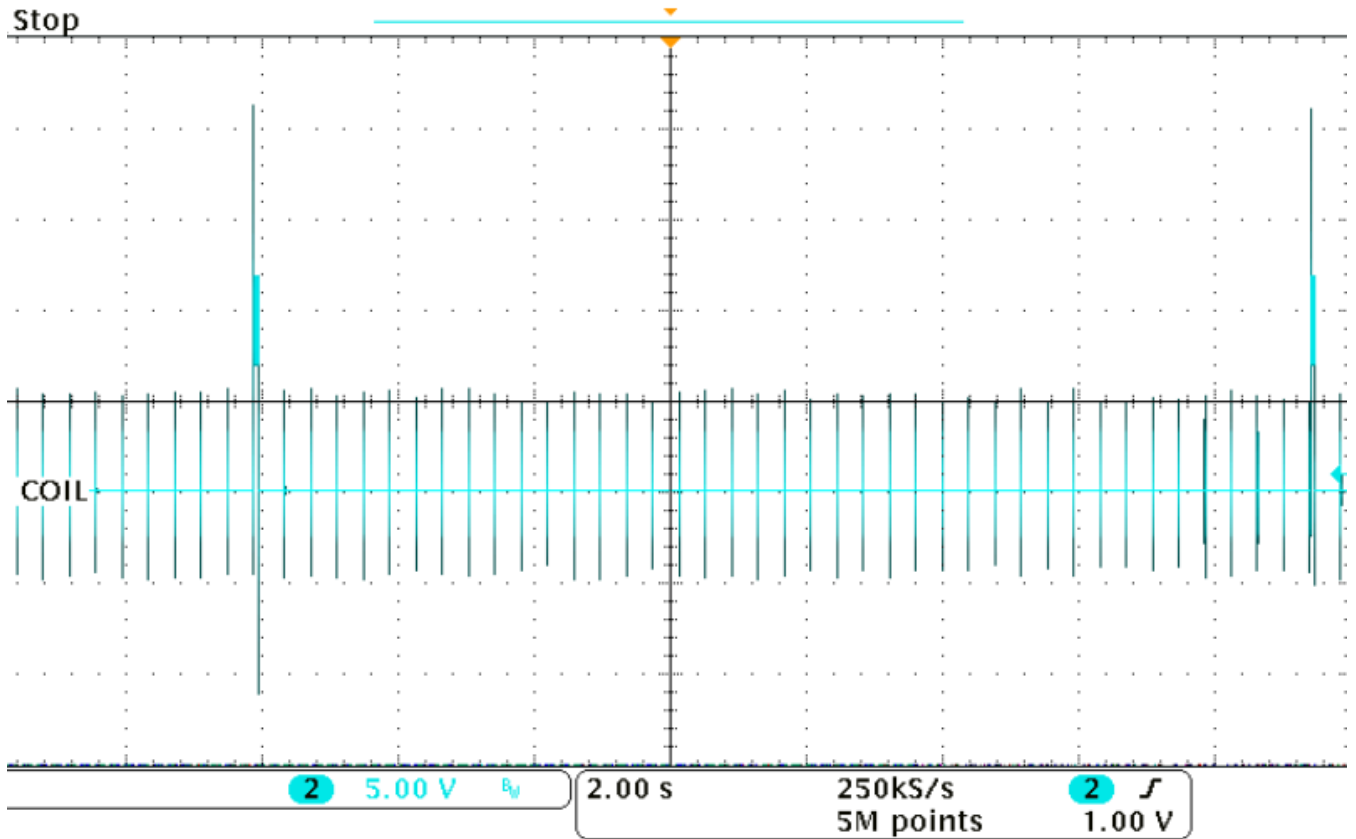
Digital Ping Every ~15sec

A digital ping every ~15 seconds is shown. This is done to "double check" that the analog ping hasn't missed a possible RX on the charger pad. If the RX seems to be taking a long time to be recognized, it can be because the analog ping is not working at its best. Check to make sure the COMM+ signal is clean. Use a known good RX unit for testing the TX prototype!

References:

1. *Unitrode Seminar Layout Guide (2004/5 Topic 4)*
2. *Layout Guidelines for Switching Power Supplies (SNVA021B)*
3. *bq500110 Datasheet*
4. *bq500210 Datasheet*
5. *bq51013 Datasheet*
6. *Ringing Reduction Techniques for NexFET High Performance MOSFETs (SLPA010)*
7. Excellent WPC compliant test tools available at the time of this writing are the "Qi-Sniffer," and the, "Qi Receiver Simulator," made by Avid Technologies (avid-tech.com). These are intended for advanced users.
8. *bqTesla Transmitter Coil Vendors* (SLUA649)

Revision History:

- Initial draft 1/5/12
- Expansion 1/18/12
- Full copy 2/22/12
- Submitted 3/6/12
- Revised 8/16/12

IMPORTANT NOTICE

Products		Applications	
Audio	www.ti.com/audio	Automotive and Transportation	www.ti.com/automotive
Amplifiers	amplifier.ti.com	Communications and Telecom	www.ti.com/communications
Data Converters	dataconverter.ti.com	Computers and Peripherals	www.ti.com/computers
DLP® Products	www.dlp.com	Consumer Electronics	www.ti.com/consumer-apps
DSP	dsp.ti.com	Energy and Lighting	www.ti.com/energy
Clocks and Timers	www.ti.com/clocks	Industrial	www.ti.com/industrial
Interface	interface.ti.com	Medical	www.ti.com/medical
Logic	logic.ti.com	Security	www.ti.com/security
Power Mgmt	power.ti.com	Space, Avionics and Defense	www.ti.com/space-avionics-defense
Microcontrollers	microcontroller.ti.com	Video and Imaging	www.ti.com/video
RFID	www.ti-rfid.com		
OMAP Mobile Processors	www.ti.com/omap	**TI E2E Community**	e2e.ti.com
Wireless Connectivity	www.ti.com/wirelessconnectivity		