bq2589x/ bq2419x / 29x

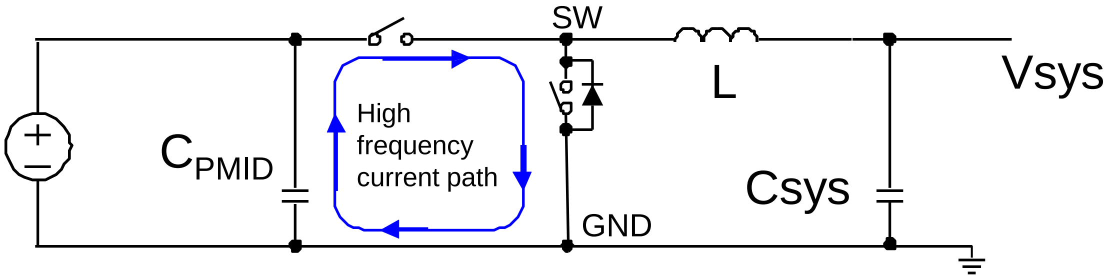
Layout Recommendations

- PCB layout guidance

1. Component placement priority

   1. Cin(VBUS & PMIC) first > 2. BTST cap > 3. REGN cap > 4. Inductor > 5. Csys > 6. C_{BAT} > 7. others for signals

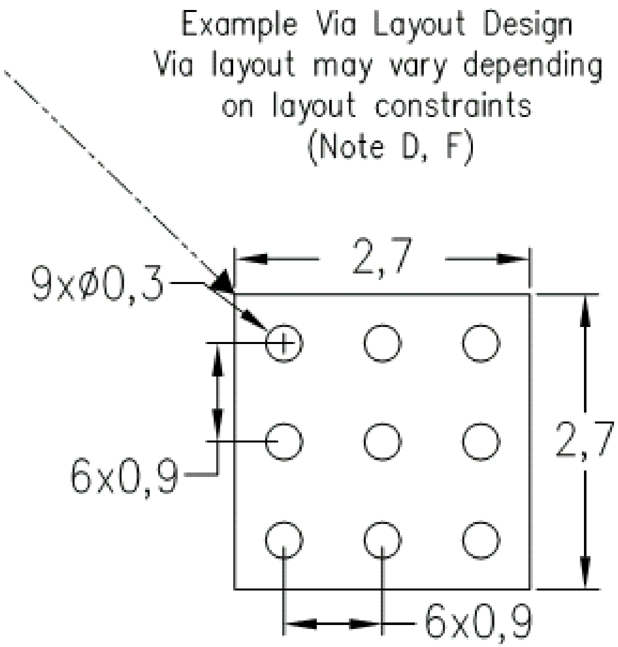
2. key consideration is to minimize high current path loop



3. Route analog ground separately from power ground. Connect analog ground and connect power ground separately. Connect analog ground and power ground together using power pad as the single ground connection point. Or using a 0Ω resistor to tie analog ground to power ground.

- PCB layout guidance

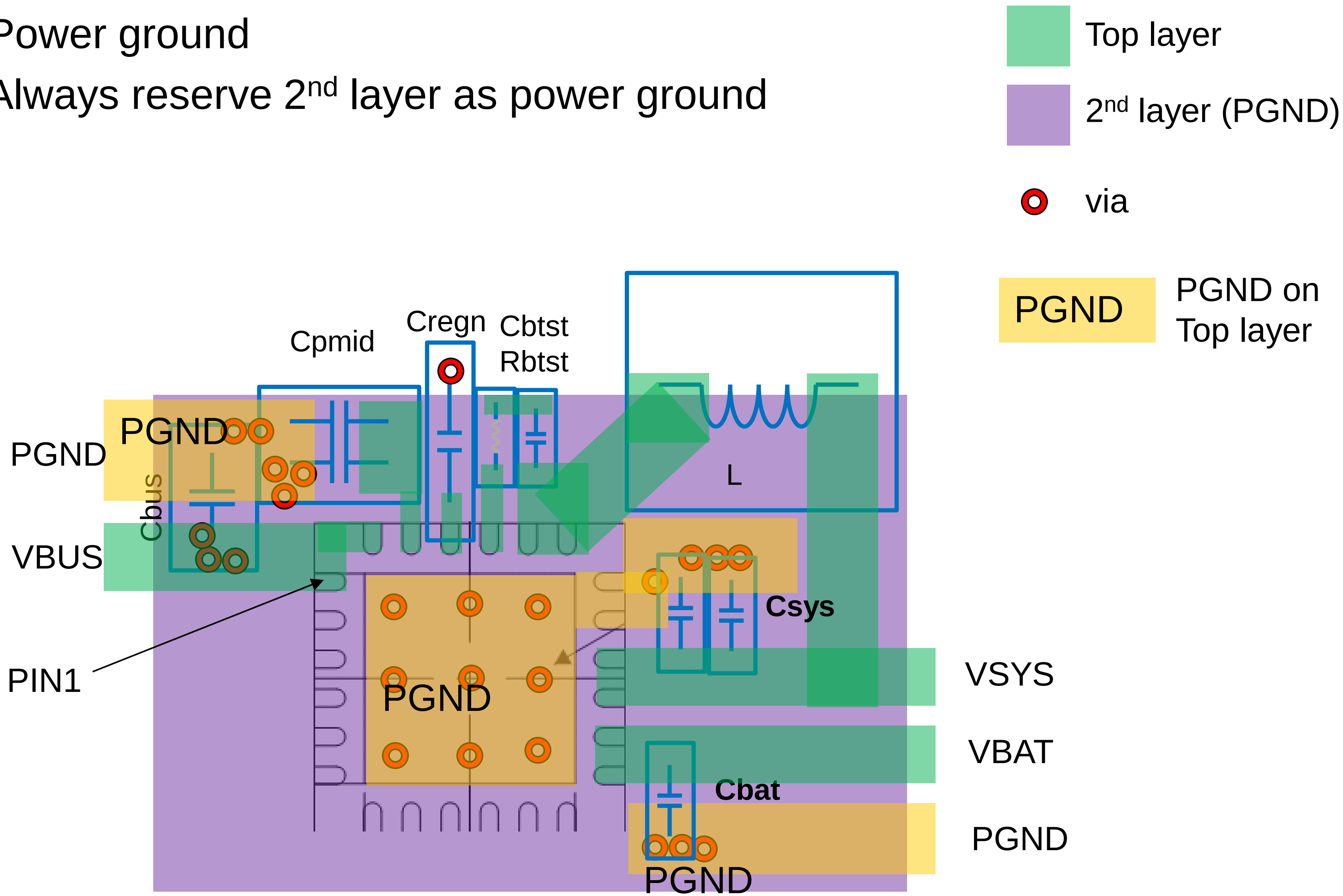
4. Thermal pad needs to follow the datasheet land pattern.



- # PCB layout guidance (layout order)

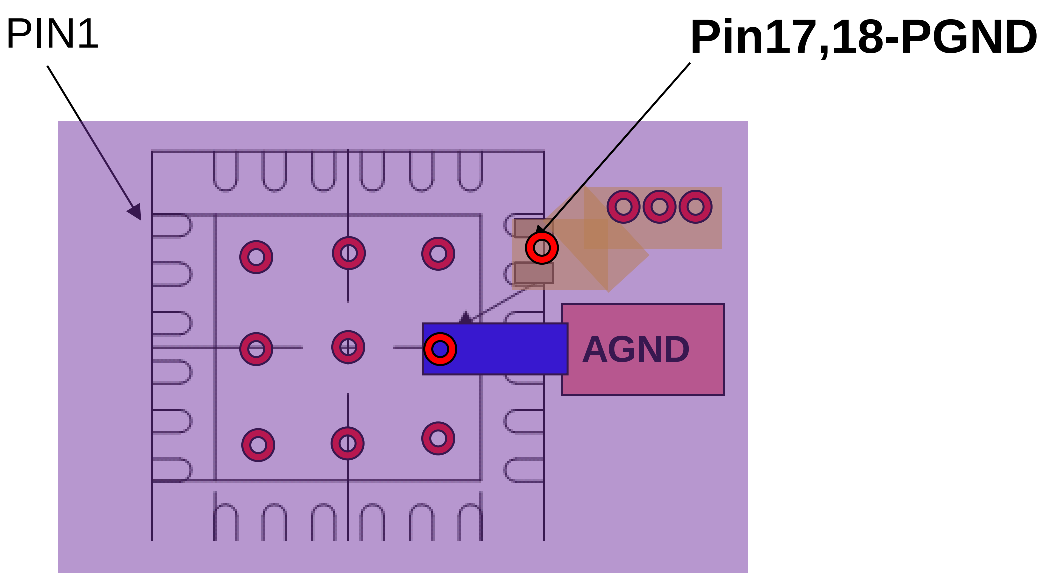
5. Power ground

- Always reserve 2nd layer as power ground



- # PCB layout guidance (layout order)

6. AGND and PGND connection



Top layer

2nd layer (PGND)

via

0Ω shunt or copper on other layer

PGND — PGND on Top layer

AGND

Connect AGND-PGND with a 0Ω shunt or copper

There is no big current flowing through the joint point. The joint point can be a thermal via or a copper pour underneath the bq2419x.

bq2589x/ bq2419x / 29x

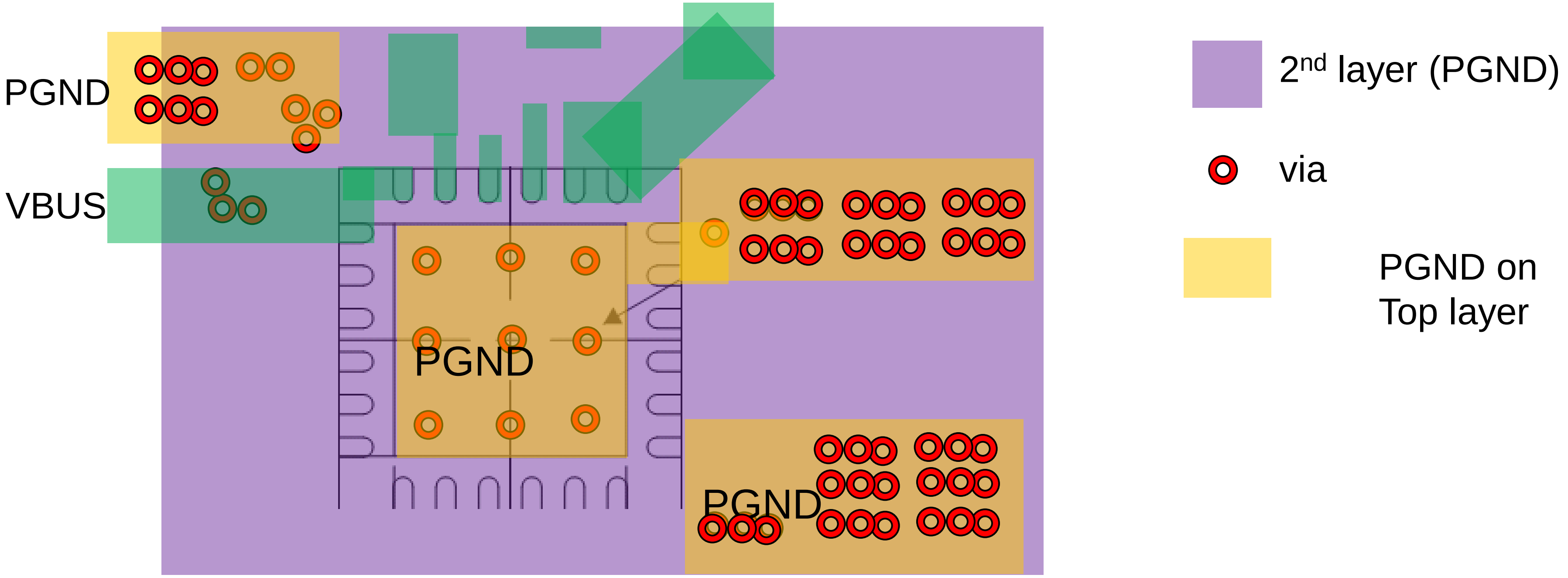
EMI Recommendations

Layout Techniques to Control EMI

1. Reduce the EMI source
2. Eliminate transmitting source
3. Trap radiated EMI

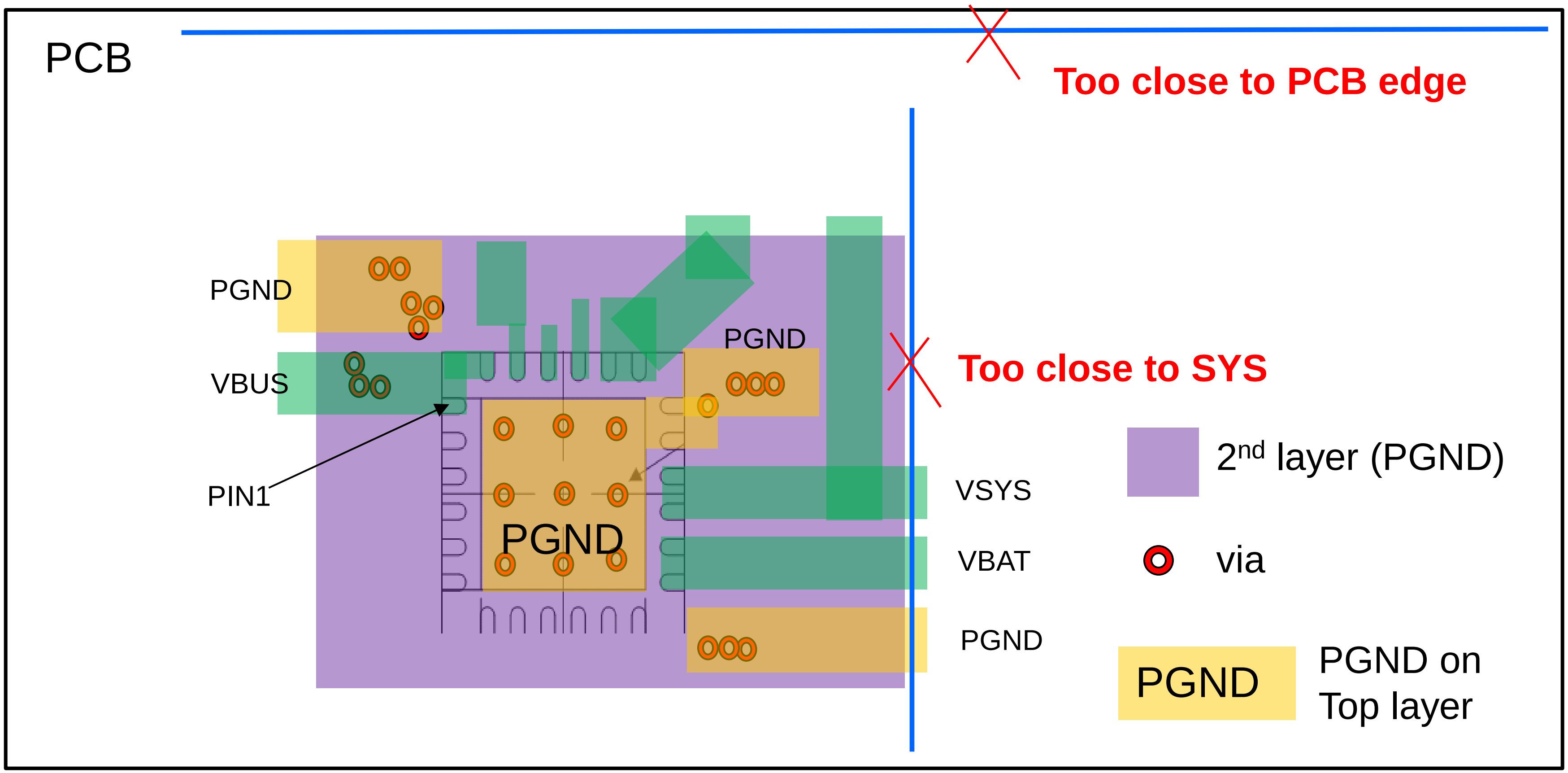
Reduce EMI Source

- Maximize continuous PGND plane on top layer to decoupling capacitors of key nodes (SW, PMID, BAT, SYS)
  - Isolated PGND plane cannot suppress EMI effectively.
- Tie PGND pin directly to PowerPad using top layer
- Use 2nd layer as PGND with plenty of VIAs to connect top and internal layer.
- Use array of VIAs to reduce VIA inductance (400nH/VIA)



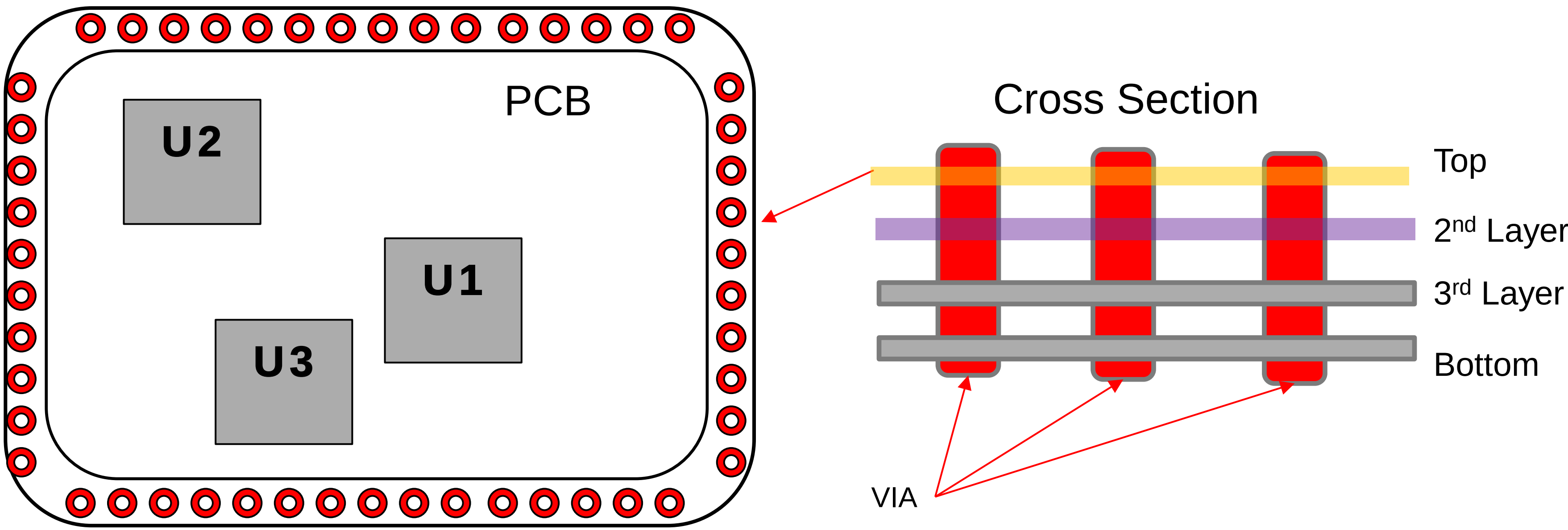
Eliminate Transmitting Source

- Avoid long traces close to PMID, SW, SYS nodes
- Avoid long traces at the edge of PCB



Trap Radiated EMI

- Use continuous PGND rings around each layer of PCB to build “faraday cage” to trap EMI leakage
- Each ring should be tied together using as much VIAs as possible to reduce PGND impedance
- No signal should be routed outside of the PGND



EMI Suppression Components

1. Add 1nF PMID, SYS, VBUS (opt.), BAT (opt.)
   - Use high frequency cap that works effectively between 100MHz and 200MHz range to return high frequency noise to ground.
   - Recommend X7R 1nF (0402) which typically has > 300M resonant frequency. Please check with vendor for impedance response.
2. Slower SW node switching slew rate
   - Resistor footprint at BTST to adjust slew rate if needed.
   - Resistor can range from 0Ω to 10Ω without performance impact. Higher resistor value (slower slew rate) can adversely affect converter efficiency.
   - Refer to “bq241xx EMI Consideration” for details

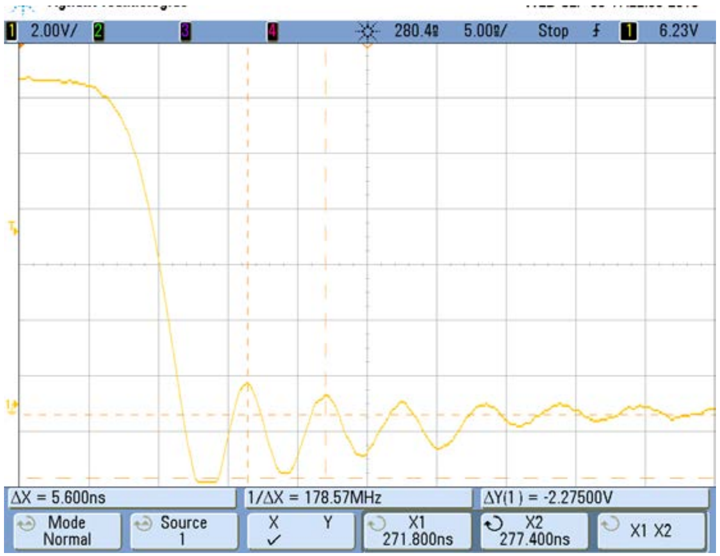
EMI Suppression Components

Advanced Techniques

1. Use Snubber to reduce EMI
   - Snubber (resistor & capacitor) footprint at SW to control EMI source
   - Components must be placed closest to SW to be effective
   - Use R=1Ω & C=15nF as reference value
2. Use EMI bead to reduce EMI
   - EMI Bead footprint at VBUS and PMID to suppress EMI
   - EMI beads selected must be able to handle high power with high impedance between 100MHz and 200MHz to effectively filter EMI at required FCC frequency range.

Effect of adding RC snubber

- RC summer on SW is an effective way to reduce the EMI. One example is on the next page with a snubber 2.7ohm and 680pF. It is a tradeoff of EMI improvement and <1% efficiency reduction.



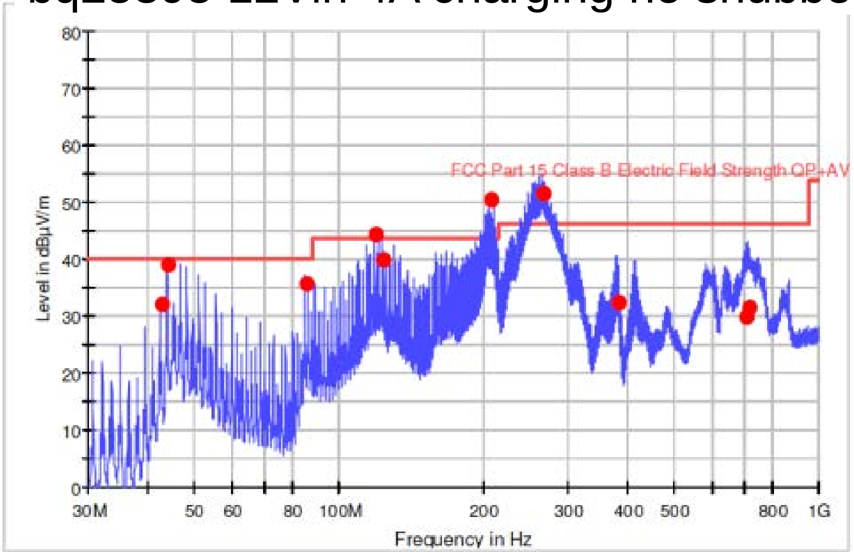
- Design of the snubber:
  Use oscilloscope with high bandwidth and find small ringing with cycle time ~3nS (~330MHz). After adding 470nF, the waveform is as 05 with 5.6ns cycle time and much higher amplitude (add cap only on SW induces high amplitude ringing). Refer to: https://www.fairchildsemi.com/application-notes/AN/AN-4162.pdf
  Another app notes: http://www.ti.com/lit/an/slva255/slva255.pdf
  The parasitic capacitance and inductance are calculated as 660pF and 1.2nH.

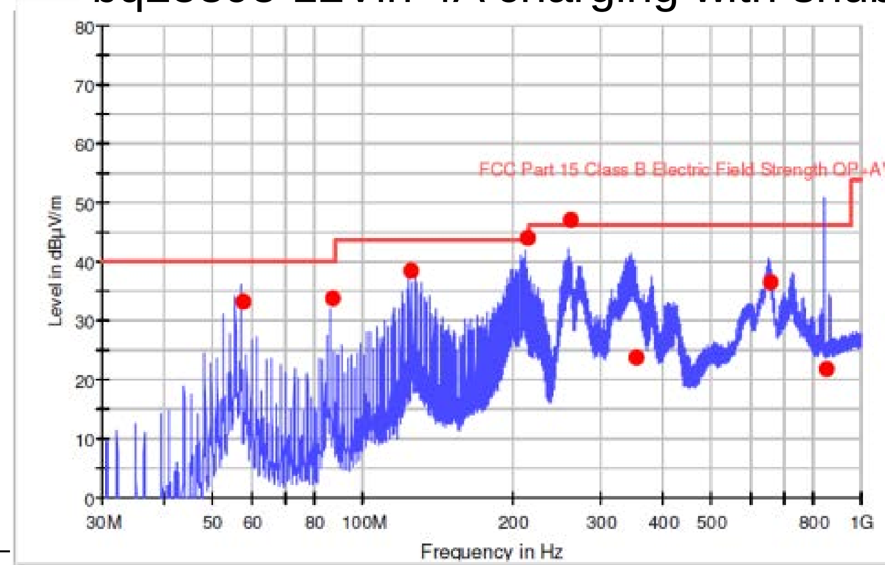
- The snubber capacitance is about 1 to 3 times of the parasitic capacitance in our tests. The snubber can be 1ohm and 2.2nF snubber. Try several different combinations to get the best results.

Effect of adding RC snubber

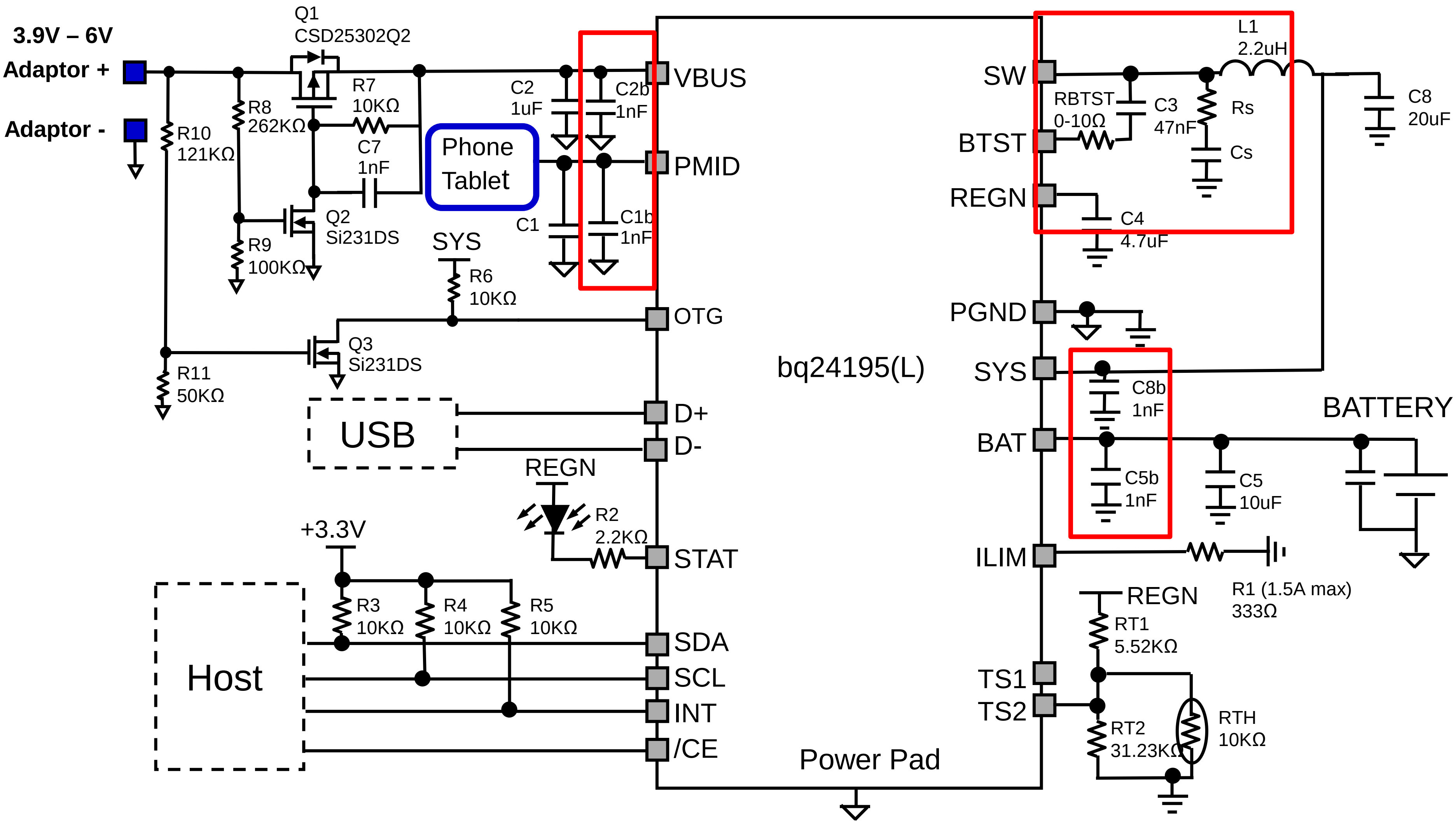
bq25895 12Vin 4A charging no snubber

bq25895 12Vin 4A charging with snubber





EMI Suppression Components



Recommended minimum C1 = 20uF (bq24195L) or 60uF (bq24195)

Cable Length Influence on EMI

- Cable can serve as an effective medium for EMI radiation, hence it is important to understand the effect of cable length on EMI
- It is important to use different cable length for EMI testing (0.5, 1, 1.5, 2 meters)
- A conductor radiates at an efficiency that is a function of the quarter wavelength of the spectral components

$$L = v / (4 \times f)$$

Where
L = Length of cable
v = Speed of light (3x10^8 m/s)
f = Frequency

Between 100MHz to 200MHz, the effective cable length for radiation is 0.375m to 0.75m.
For 5m cable, the effective frequency is 15MHz

- A short cable (0.375m to 0.75m) radiates more effectively than longer cable (5m) since the shorter cable is within test limit.

* Source: Intel – EMI Design Guidelines for USB components