

MC13892 Layout Guidelines

By: Jorge Ramirez

1 Purpose

This document is intended to show good practices on how to layout the MC13892 PCB for a correct functionality of the whole system.

2 Scope

This document contains the packaging and recommended footprint for the IC, pinout, and layer stacking recommendations, and layout tips for routing transmission lines and switching power supply traces.

3 Packaging

The MC13892 is offered in two BGA packages: a 139 pin 7x7 mm, 0.5 mm pitch package, and a 186 pin 12x12 mm, 0.8 mm pitch package. The package style is a low profile BGA semi populated matrix, MC13892VK includes 139 balls including 4 sets of triple corner balls, so in total 131 assigned signal pins, where the MC13892VL includes 186 balls, all of which are assigned signal pins.

Contents

1 Purpose	1
2 Scope	1
3 Packaging	1
4 Pin Escape	7
5 SPI/I2C Communication and Real Time Clock Signals	9
6 Switching Power Supplies Traces	10
7 Feedback Signals	13
8 LDO and Battery Charger Routing	14
9 References	15

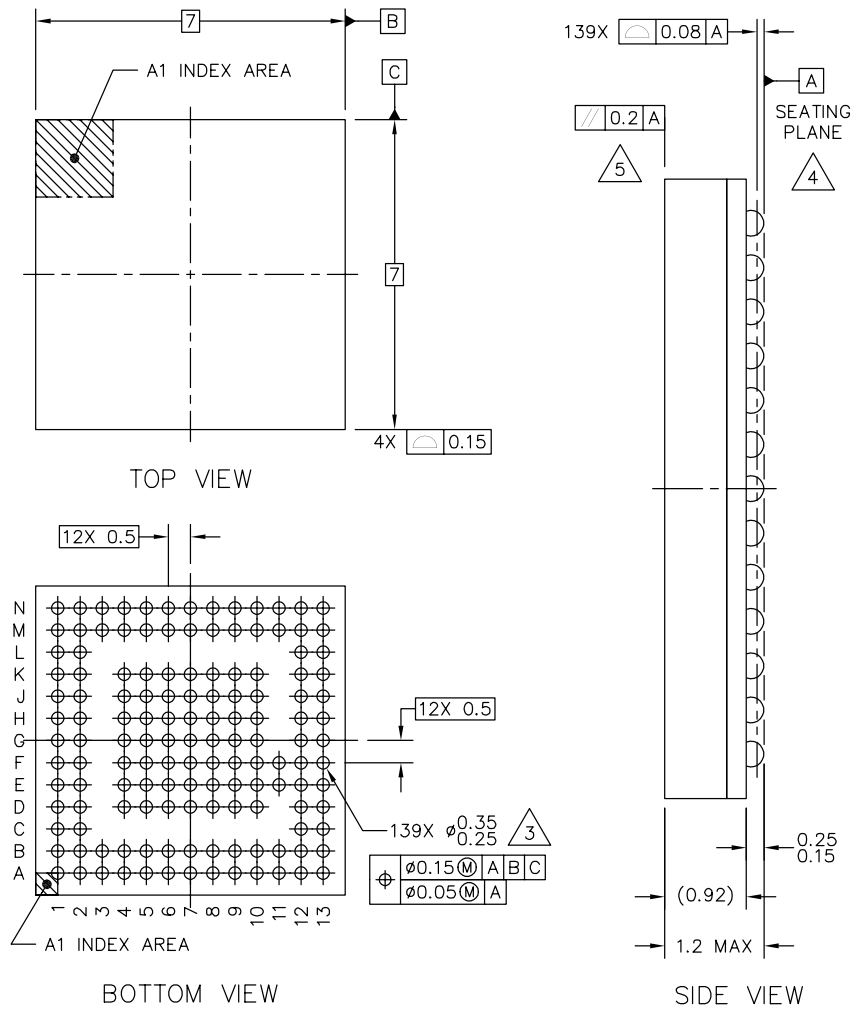


Figure 1. MC13892JVK Package

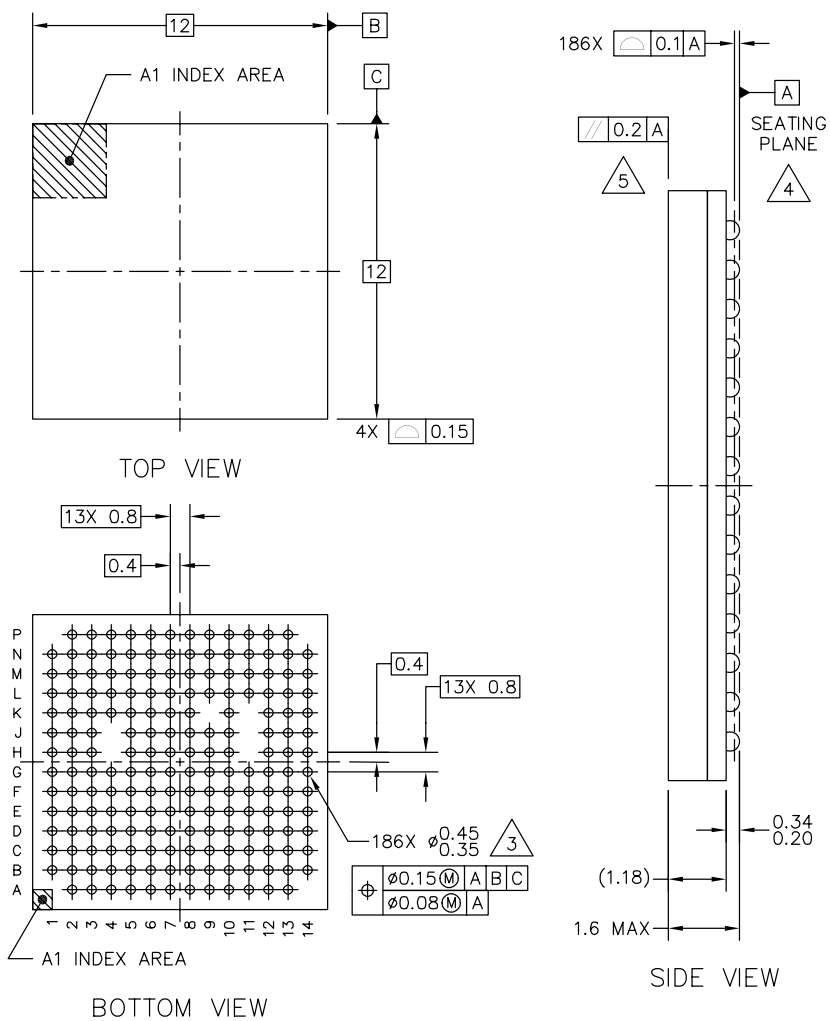


Figure 2. MC13892JVL Package

3.1 Recommended Footprints

The MC13892JVK footprint consists of 10 mil pads with a 19.68 mil pitch, and the MC13892JVL footprint has 16 mil pads with a 31.496mil pitch. These are recommended as shown in [Figures 3](#) through [4](#).

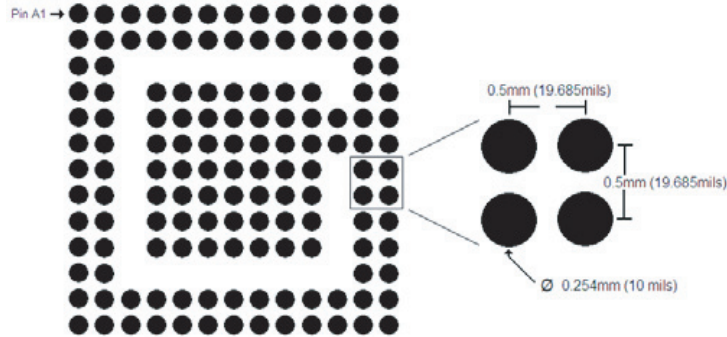


Figure 3. Recommended Footprint for the MC13892JVK (Top View)

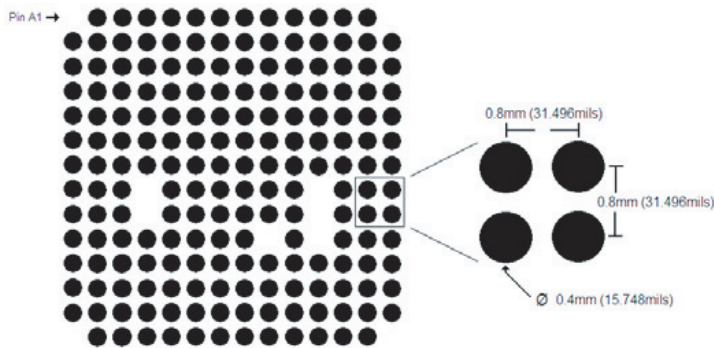


Figure 4. Recommended Footprint for the MC13892JVL (Top View)

A	VUSE2	VUSE2	VINUSE2	SWBSTIN	GND5ABST	GND5BL	NC	MODE	VCORE	BATT	CHRGRAW	CHRGCTRL2	CHRGCTRL2
B	VUSE2	GP01	DVS2	SWBSTOUT	LEDB	LEDP	LEDR	GND5CORE	V5COREDIG	BP	CHRGCTRL1	BATT5NSCC	CHRGCTRL2
C	VINFL	V5DDR									CHRG5NS	BATT5NS	
D	VUSB	VSD		SWBSTFB	LEDMD	DVS1	REFCORE	CHRG5E1B	UCCELL	BATT5FET		B5PNS	PWR0N1
E	UMBUS	VRL		LEDG	GND5LED	UD	PUMS2	GND5HRG	CHRG5LED	PWR0N2	ADTRG	INT	GND5M1
F	GND5M6	V5BUSEN		SW5FB	LEDAD	GND5SUB	GND5SLB	GND5SLB	GP03	GP02	RESETB5M0J	RESETB	SW5OUT
G	SW5OUT	VIN5EB		SW5FB	GND5REG2	GND5SUB	GND5SLB	GND5SLB	PUMS1	VDI		GP04	SW5IN
H	SW5IN	M5SO			GND5P1	GND5REG3	GND5SUB	GND5SLB	GND5SLB	GND5CTRL	SW5FB	STANDB5SEC	SW5M2N
J	SW5IN	M5SI		CLK52M0J	STANDB5	GND5ADC	GND5REG1	PWR0N3	TSX1	SW5FB		TSX2	SW5ZOUT
K	SW5OUT	SP5VCC		PWG5TRV1	CLK53K	V5CAM	CFP	CFM	AD5N5	AD5N6		W5DE0DRV	GND5M2
L	GND5M4	CS										TSY2	W5DE0
M	V5GEN3	CLK	V5GEN2	V5SR5C	GND5RTC	V5NCA5DRV	PWG5TRV2	V5DIG	V5IND5G	V5GEN5DRV	AD5N7	TSY1	TS5REF
N	V5GEN3	V5GEN3	V5GEN5DRV	V5GEN5DRV	XTAL2	XTAL1	V5VALD5IO	V5VALD5IO	V5V5OH	V5VIN5CH	V5GEN1	TS5REF	TS5REF

	Regulators
	Switchers
	Backlights
	Control Logic
	Charger
	RTC
	Grounds
	USB
	ADC
	SPI/I2C
	No Connect

Figure 5. MC13892JVK Pinout

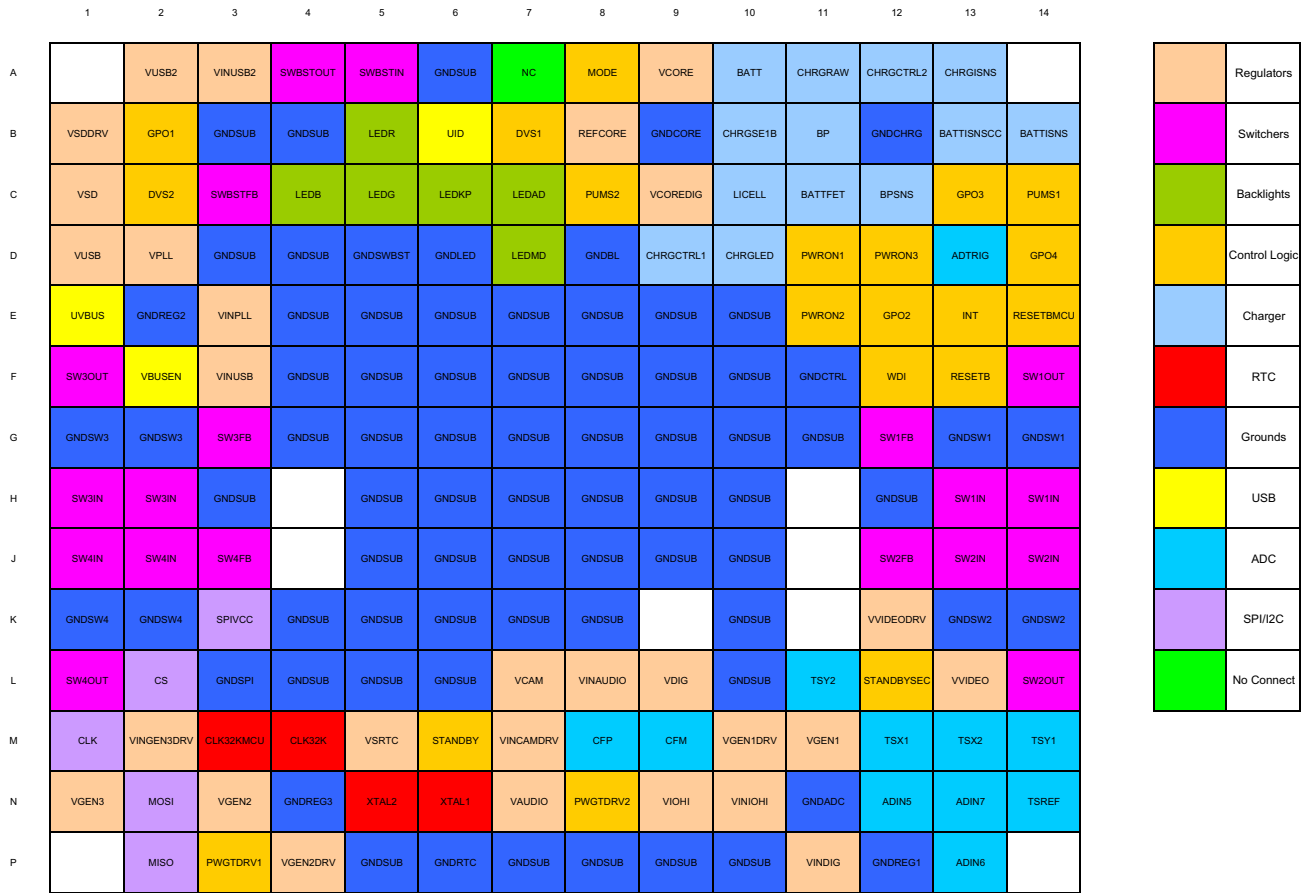


Figure 6. MC13892JVL Pinout

Signals are placed on the most convenient sides and grouped according to which block they belong. The most critical signals are located on the outer sides of the integrated circuit to easily route them, which improves the performance of the whole circuit.

Table 1 shows the recommended layer stack-up for the signals to receive good shielding.

Table 1. Layer stacking recommendations

Layer	Stack-up
Layer 1 (Top)	Signal
Layer 2 (Inner 1)	Fanout/Ground
Layer 3 (Inner 2)	Power
Layer 4 (Bottom)	Signal/Ground

NOTE

Note: A more detailed layer design may be required to route the i.MX. If the MC13892 is being interfaced with an i.MX, just four of the layers will be needed to route it. Please note, for the MC13892VK, the fan-out needs to be made on the layer right next to the top, since micro vias will be needed to take the signals out of the IC as explained in the Pin Escape section.

3.2 Component Placement Hints

- Place these components first so they are as close as possible to the IC:
 - Place input caps of the switchers first (SW1, SW2, SW3, and SW4)
 - Place the output diode of the boost (SWBST)
 - Place output caps and diodes of the boost (SWBST) and SWLED
 - Place the REFCORE cap
- Shield feedback paths of the switchers (trace them on the bottom so the ground and power planes shield these traces).
- Sense pins must be directly connected to the pads of the sense resistors via separate traces (BATT, BATTISNS, BATTISNSCC, BPSNS, and CHRGISNS).
- BATTISNSCC must be connected directly to the pads of the sense resistor via a separate trace from BATTISNS.
- External pass devices of regulators must be closer to the load, but be careful with the sense and drive traces.
- Output caps of LDOs with the external pass device option should be close to the transistor (VVIDEO, VCAM, VSD, VGEN1, VGEN2 and VGEN3).
- Avoid a coupling trace between important signal/low noise supplies (like REFCORE) from any switching node.
- Ensure each of the components is referenced, or at least it does not have a long return path to the ground of the block to which it belongs.
- Trace REFCORE away from, or shielded from SWLEDOUT
- Traces that go from BP to a pin of the MC13892 must reach a capacitor before the pin.
- Routes for GNDSW should be suitably large. They will carry heavy switching currents.
- Switcher inductors should also be placed close to the IC, and switching node traces should be short and wide, to reduce conduction losses.

4 Pin Escape

Pin Escape is defined as the manner by which the signals leaving the IC from their associated pin, can be delivered on a circuit board trace of a specified size, to a point at which the traces can acquire the size necessary to perform their process function.

4.1 The MC13892VK Package

The most convenient way to approach pin escape is to take the signals coming from the bottom of the IC, with 3.0 mil traces on the top layer. Micro vias can be put on the pads of the footprint for the signals on the center of the IC. These will take the signals from the outer layer to the following one, once there, since this layer will not be as crowded as the top one, 5.0 mil traces can take the signal between the vias. These traces can become bigger once they are out from the bottom of the IC. More micro vias can be put among the pins to take the signal to another layer.

The recommendation is for the micro vias to have a 10 mil diameter (same as the pads of the footprint) with a 5.0 mil drill, which along with the 3.0 mil traces, will result in a 3.0 mil separation between the pad and the trace. This can be done with a 0.5 oz copper thickness on the top layer. The rest of the layers can have 1.0 oz copper thickness for better current handling. Care must be taken on these layers that the separation is more than 5.0 mils between traces, vias, pads, and planes. Reference [Figure 7](#).

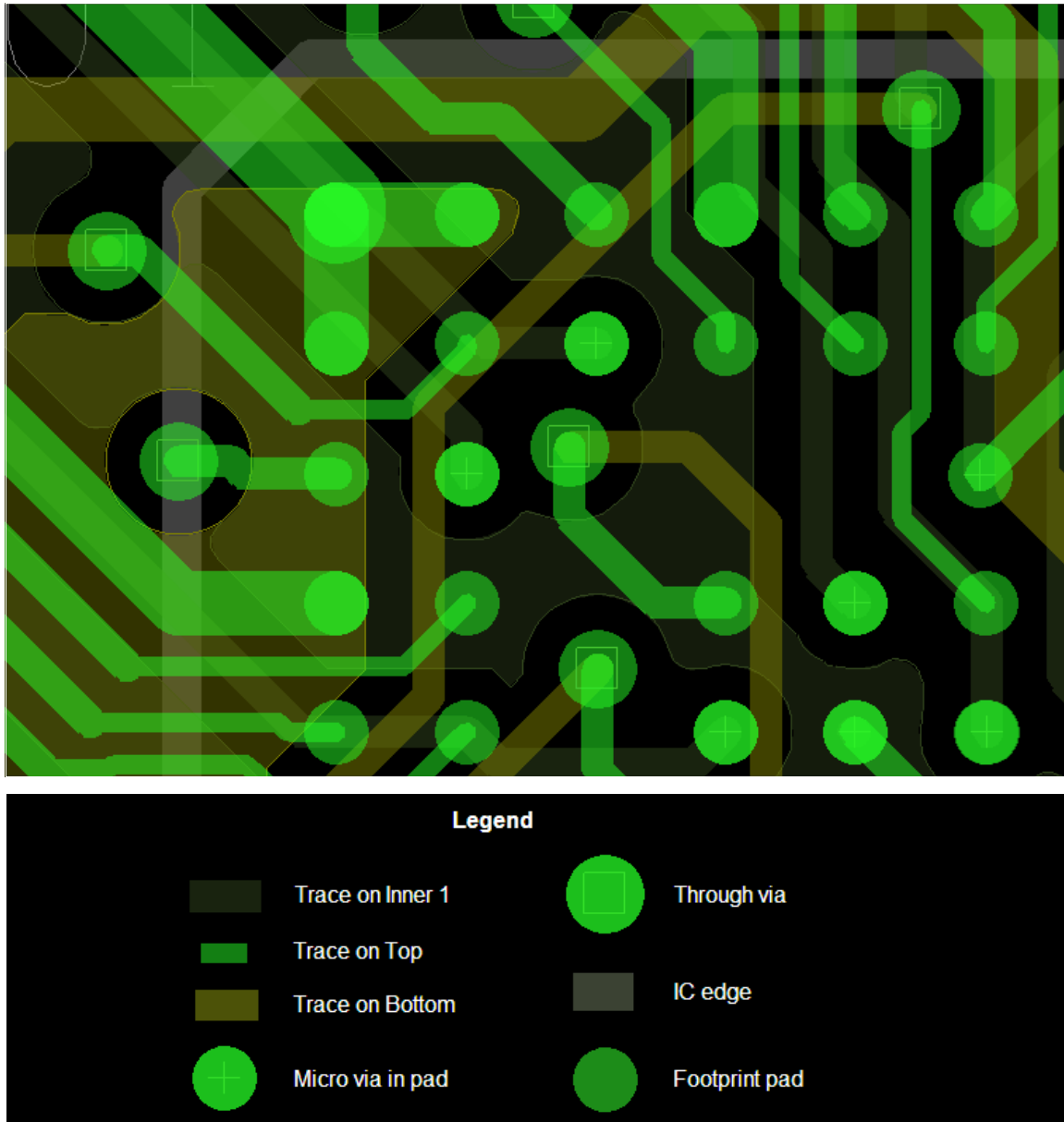


Figure 7. Pin Escape for the MC13892VK

[Figure 7](#) shows the top left side of the MC13892VK. Note how the outer pins can have wider traces, and inner pins have narrow traces that become wider once out (escaping) from the bottom of the IC.

Bigger vias can be put on the clear area of rows C and L, and columns 3 and 11 (see [Figure 5](#)) to route critical signals.

The most convenient way to trace a signal depends on the length and width it must have. Tips for each kind of signal are found in [SPI/I2C Communication and Real Time Clock Signals](#) and [Switching Power Supplies Traces](#).

4.2 The MC13892VL Package

Since the pins of this version of the device are wider, the use of micro vias can be avoided, and through vias can be used for the whole design for cost savings. The problem with through vias is that, since they go through the entire

board from top to bottom, the drill can not be too narrow. The MC13892VK's small pins do not allow their use, but MC13892VL does.

Vias placed on the pads of the footprint can have a 12 mil diameter hole, with an 8.0 mil drill for the manufacturer to be able to use a conductive fill in them. 12 mil diameter vias will allow 9.0 mil traces to pass through them on the inner and bottom layers of the board, while 16 mil pads of the footprint will only allow 5.0 mil traces on the top layer below the IC. It is recommended to place high current traces on the bottom layer.

5.0 mil traces passing through 16 mil pads will result in a 5.0 mil separation between the pad and the trace, which can be made with a 1.0 oz. copper thickness.

Taking advantage of the through vias, the fanout can be primarily made on the top and bottom layers, for the inner ones to be completely ground and power.

5 SPI/I2C Communication and Real Time Clock Signals

CLK is the fastest signal of the system, so it must be given special care. Here are some tips for routing the communication signals:

To avoid contamination of these delicate signals by nearby high power or high frequency signals, it is a good practice to shield them with ground planes placed on adjacent layers. Make sure the ground plane is uniform throughout the whole signal trace length.

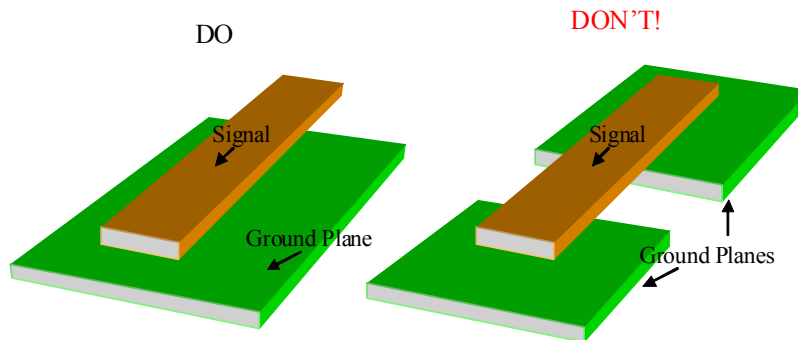


Figure 8. Recommended Shielding for Critical Signals.

These signals can be placed on an outer layer of the board to reduce their capacitance in respect to the ground plane.

The crystal connected to pins XTAL1 and XTAL2 must not have a ground plane directly below.

The following are clock signals: CLK, CLK32K, CLK32KMCU, XTAL1, and XTAL2. These signals must not run parallel to each other, or in the same routing layer. If it is necessary to run clock signals parallel to each other, or parallel to any other signal, then follow a MAX PARALLEL rule as follows:

- Up to 1 inch parallel length – 25 mil minimum separation.
- Up to 2 inch parallel length – 50 mil minimum separation.
- Up to 3 inch parallel length – 100 mil minimum separation.
- Up to 4 inch parallel length – 250 mil minimum separation.

Care must be taken with these signals not to contaminate analog signals, as they are high frequency signals. Another good practice is to trace them perpendicularly on different layers so there is a minimum area of proximity between signals.

6 Switching Power Supplies Traces

6.1 Buck Converter

To place and route adequately external switcher components, follow the current paths of a Buck converter, as shown in [Figure 9](#) and [Figure 10](#).

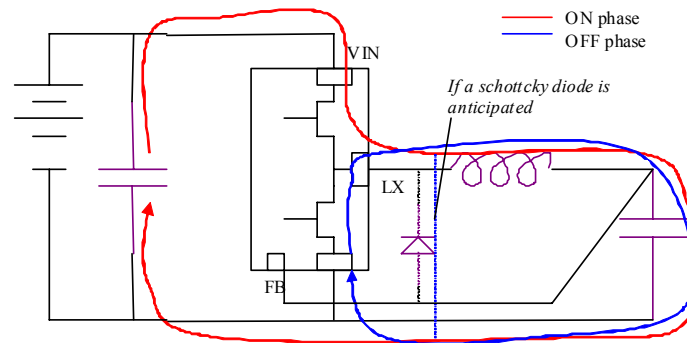


Figure 9. Current flow on a buck converter.

There are paths of 2 colors and paths with only one-color. Special attention must be paid to the one-color paths, because there the current alternates between zero and full value. These one-color paths are areas with high di/dt , that generate a significant magnetic field around the PCB traces.

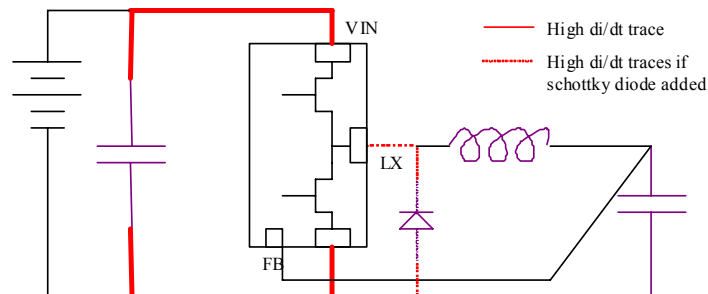


Figure 10. Critical traces of a buck converter.

It is important that all of these critical traces must be kept as short as possible. Each trace has an inductance proportional to the length. Inductance cannot tolerate high di/dt , so a high di/dt in long traces will result in a high ringing dv/dt . A wide trace is not a compensating solution for a long trace, as inductance is NOT inversely proportional to the width of the trace.

Traces must be very short and fairly wide (large amount of current flow in these traces), and should not go through any vias, as they also add impedance and inductance. Switcher components such as inductors and capacitors should be as close to the IC as possible to achieve a proper trace length.

Make sure that each trace is capable of handling the current it will carry. As a rule of thumb, it can be considered that a 10 mil trace with a thickness of 1.0 oz/ft², is capable of handling 1.0 ampere. Therefore, a trace that will carry 1.3 A in a 1.0 oz/ft² copper layer (as could be the case with an SW1 signal, because of the transient off the caps), must be 13 mils wide.

The switcher output to output capacitor connections should introduce negligible parasitic inductance, with regard to the coil inductance. Nevertheless, the switcher output capacitor to load connections are critical traces with high di/dt , and must have a maximum parasitic inductance of 1.0 nH, and an ohmic impedance of less than 20 m Ω .

6.2 Input Capacitor

An input capacitor is recommended to ensure a stable input voltage of the switcher during large load transients. It will provide the necessary energy to source current to the switcher until the battery supply is able to fill the demand. Ceramic capacitors fit well into applications, as they have a very small ESR.

The placement of this capacitor on the PCB is very important.

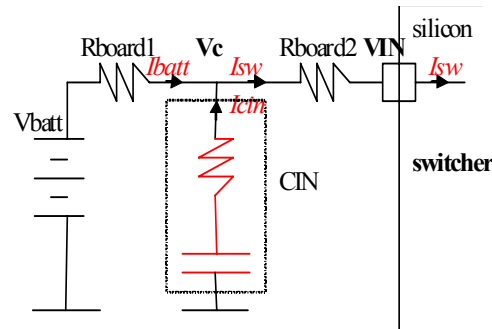


Figure 11. Input Capacitor Diagram.

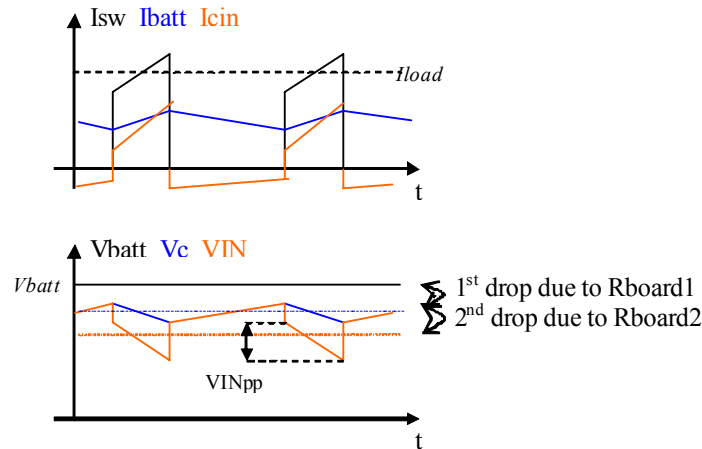


Figure 12. Current and Voltage Signals on a Buck Converter.

Figures 11 and 12 (ESR effect of CIN disregarded) bring out the effect of a resistive path Rboard1 and Rboard2, that must be minimized as much as possible. Rboard2 must be kept as small as possible, as all the switcher current (transient steady state and also local transient peaks) flows through this resistive path. Place CIN ACAP (as close as possible) to the chip.

Depending board constraints, it may not be possible to put the switchers close to CIN. In this case, it is recommendable to add an extra, small capacitor C_{in_sw} (still ceramic) ACAP to the switcher input. This extra cap will provide the main transient current during crossover transition of the switcher, whereas CIN will continue to prevent a drop on the input supply during large load transients.

Note: The recommended value for this capacitor on this application is 4.7 μ F.

6.3 Switching Node

The components associated with this node must be placed as close as possible to one another to assure the switching currents are small enough not to contaminate other signals. However, care must be taken to ensure the copper traces joining these components together on this node are capable of handling the necessary current.

Figure 13 shows the placement and layout philosophy for a recommended PCB:

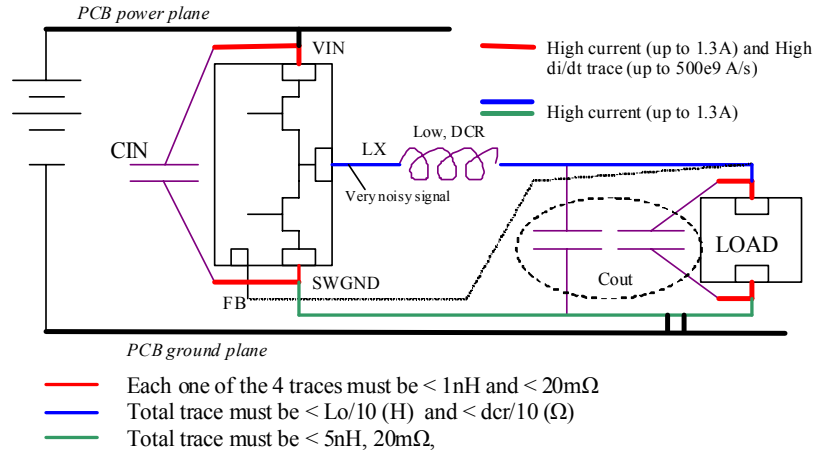


Figure 13. Placement and Layout Philosophy for a Recommended PCB.

6.4 Boost converter

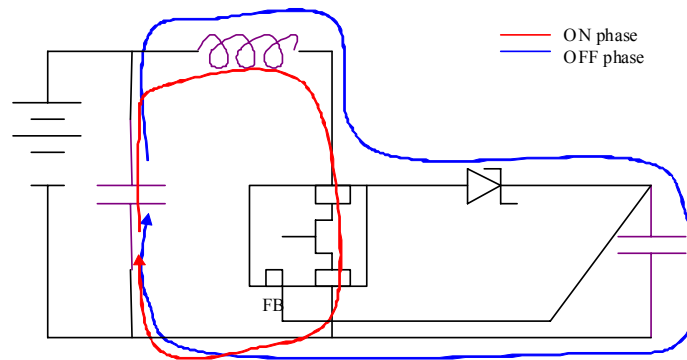


Figure 14. Current Flow on a Boost Converter.

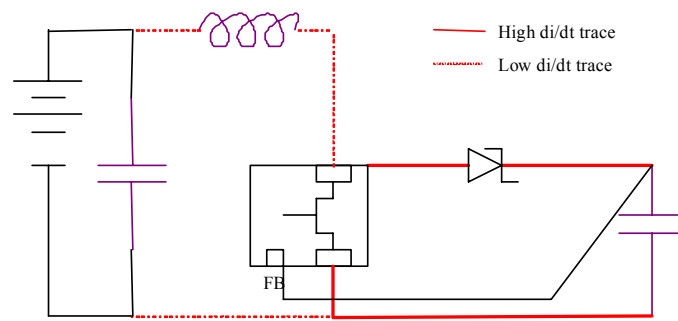


Figure 15. Critical Traces on a Boost Converter.

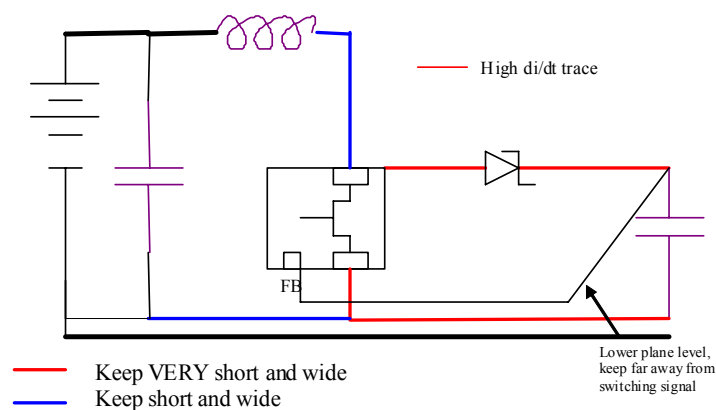


Figure 16. Placement and Layout Philosophy for a Recommended PCB.

7 Feedback Signals

To compensate for the voltage drop on the MC13892 power management IC switcher's line due to the PCB wire resistance, the PCB wire resistance needs to be placed inside the closed loop of the switcher. The goal is to stabilize the voltage on that switcher line output.

The capacitor and coil components L and C are usually placed close to the MC13892 package. The voltage, SWxOUT, is equal to the programmed output switcher voltage, minus the dropout caused by the PCB trace. This dropout is also dependent on the DC load current on SWxOUT. In a typical configuration, the feedback loop is not usually shielded.

The feedback signal is the core of the loop, with the functional information fed back and compared for reference (this trace must be shielded from other signals). Moreover, it is a very high-impedance signal, so keep its trace thin and far from the switching signal (the length of this trace does not matter).

Freescale recommends following the L and C configuration as designated. The output capacitor should be placed near the point where the current is drawn on the output side of the PCB wire resistance, as shown in [Figure 14](#). The PCB wire resistance is now inside the closed loop of the switcher, so it does not impact the DC voltage at SWxOUT.

The switcher feedback trace must be shielded, as this line is attached to a high-impedance point in the MC13892. Any perturbation on this line must be minimized to ensure the stability of the switcher.

[Figure 17](#) shows the recommended configuration for the MC13892 power management IC, to stabilize the voltage on the switcher line.

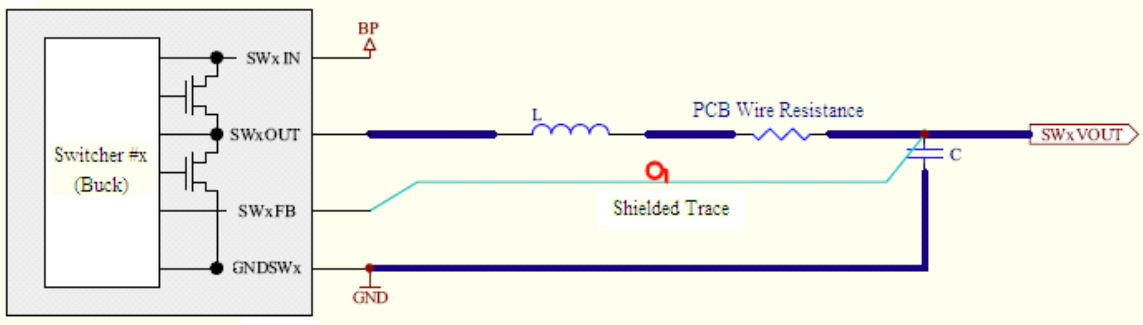


Figure 17. Recommended Configuration.

8 LDO and Battery Charger Routing

The LDOs and battery charger do not handle much current in this application, so their routing is not critical. The same philosophy must be followed for current handling on the traces though.

For a proper functionality of these supplies, bypass and output capacitors must be placed near the IC to achieve a good width and shortness of the traces. The ESR of the output capacitors of the external PNP LDOs must be between 20 and 100 mΩ, in order to achieve good stability. If they are not within these values, an additional resistor will be needed. This resistance can be achieved with a copper trace on the board and is also a cost saving.

The critical signals for the charger that must not be contaminated by high frequency or power ones are highlighted in [Figure 18](#).

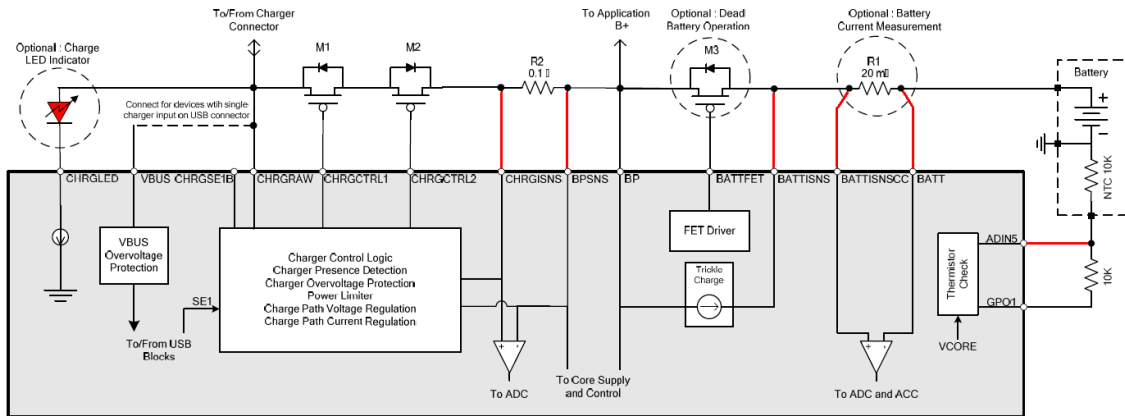


Figure 18. Critical Signals on the Battery Charger.

The sense lines should be routed from the sense resistor endpoints and should not be included in high current charge path. They should also have a maximum of 1.0 nH parasitic inductance and 30 mΩ of resistance.

Charge path can handle up to 1600 mA, so following the rule listed previously, for 1.0 oz copper, the trace width must be at least 16 mils.

As a placement suggestion, the thermistor must be located as close to the battery as possible for a correct temperature reading.

9 References

- MC13892 Data sheet

How to Reach Us:

Home Page:

www.freescale.com

Web Support:

<http://www.freescale.com/support>

USA/Europe or Locations Not Listed:

Freescale Semiconductor, Inc.
Technical Information Center, EL516
2100 East Elliot Road
Tempe, Arizona 85284
1-800-521-6274 or +1-480-768-2130
www.freescale.com/support

Europe, Middle East, and Africa:

Freescale Halbleiter Deutschland GmbH
Technical Information Center
Schatzbogen 7
81829 Muenchen, Germany
+44 1296 380 456 (English)
+46 8 52200080 (English)
+49 89 92103 559 (German)
+33 1 69 35 48 48 (French)
www.freescale.com/support

Japan:

Freescale Semiconductor Japan Ltd.
Headquarters
ARCO Tower 15F
1-8-1, Shimo-Meguro, Meguro-ku,
Tokyo 153-0064
Japan
0120 191014 or +81 3 5437 9125
support.japan@freescale.com

Asia/Pacific:

Freescale Semiconductor China Ltd.
Exchange Building 23F
No. 118 Jianguo Road
Chaoyang District
Beijing 100022
China
+86 10 5879 8000
support.asia@freescale.com

For Literature Requests Only:

Freescale Semiconductor Literature Distribution Center
P.O. Box 5405
Denver, Colorado 80217
1-800-441-2447 or +1-303-675-2140
Fax: +1-303-675-2150
LDCForFreescaleSemiconductor@hibbertgroup.com

Information in this document is provided solely to enable system and software implementers to use Freescale Semiconductor products. There are no express or implied copyright licenses granted hereunder to design or fabricate any integrated circuits or integrated circuits based on the information in this document.

Freescale Semiconductor reserves the right to make changes without further notice to any products herein. Freescale Semiconductor makes no warranty, representation or guarantee regarding the suitability of its products for any particular purpose, nor does Freescale Semiconductor assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation consequential or incidental damages. "Typical" parameters that may be provided in Freescale Semiconductor data sheets and/or specifications can and do vary in different applications and actual performance may vary over time. All operating parameters, including "Typicals", must be validated for each customer application by customer's technical experts. Freescale Semiconductor does not convey any license under its patent rights nor the rights of others. Freescale Semiconductor products are not designed, intended, or authorized for use as components in systems intended for surgical implant into the body, or other applications intended to support or sustain life, or for any other application in which the failure of the Freescale Semiconductor product could create a situation where personal injury or death may occur. Should Buyer purchase or use Freescale Semiconductor products for any such unintended or unauthorized application, Buyer shall indemnify and hold Freescale Semiconductor and its officers, employees, subsidiaries, affiliates, and distributors harmless against all claims, costs, damages, and expenses, and reasonable attorney fees arising out of, directly or indirectly, any claim of personal injury or death associated with such unintended or unauthorized use, even if such claim alleges that Freescale Semiconductor was negligent regarding the design or manufacture of the part.



Freescale™ and the Freescale logo are trademarks of Freescale Semiconductor, Inc. All other product or service names are the property of their respective owners.

© Freescale Semiconductor, Inc. 2010. All rights reserved.

AN3964
Rev. 1.0
3/2010