SILICON LABS

AN1343: RS9116 B00 Board Layout Guidelines

Version 1.0

August 20, 2021

Table of Contents

1 Introduction 3
2 Placement Guidelines 4
3 RF Layout Guidelines 6
4 SDIO/SPI Layout Guidelines 8
5 USB Layout Guidelines 10
6 UART Layout Guidelines 11
7 Power Supply Layout Guidelines 12
8 GND Pour Layout Guidelines 16

1 Introduction

This Application Note provides PCB layout guidelines for the RS9116 B00 SiP module. These guidelines cover parts placement, various critical traces routing like RF, host interfaces routing like SDIO/SPI, USB, UART, power routing, and GND pour. B00 module placement and routing can be done within a 4-layer PCB stack-up. However, the designer can choose higher number of layers, based on product needs. This Application Note describes placement and routing considering 4-layer stack-up.

Below is the recommended layer stack-up and placement/routing considering a 4-layer stack-up. Exact PCB thickness can be as per the layer stack-up provided by the PCB manufacturer. It is recommended to use a PCB thickness close to 1.6 mm, but the designer can choose suitable thickness based on product needs.

- Layer 1 : Parts Placement, RF circuitry, Interface signals (SDIO/SPI, USB, UART)
- Layer 2 : GND
- Layer 3 : Power supply star routing
- Layer 4 : Power supply star routing, Interface signals (SDIO/SPI, UART) and few GPIOs

2 Placement Guidelines

The designer can choose a suitable antenna as per the product needs, like Chip, on-board PCB trace, external PCB trace, Dipole, etc. In this Application Note, a Chip antenna is considered on the RF_PORT1 pin of the module. However, the designer can choose any antenna based on product needs.

Here are the recommendations for the placement of the B00 module and its associated parts. Parts are placed on the same side of the B00 module, in this document. However, the designer can choose to place them on either side of PCB for achieving best placement and routing. The steps below can be sequentially followed to achieve optimal placement.

1. To get the best RF performance from the B00 circuitry and antenna, it is recommended to place antenna circuitry near the PCB edge. Antenna part guidelines must be followed for its placement and routing.
2. Place RF components, antenna and the B00 module on the same side of the PCB, so that there are no vias on RF traces.
3. The RF front-end components and the B00 module should be placed with minimal distance from the RF_PORT1 pin on the module till the antenna, so that RF traces lengths are as short as possible.
4. Decoupling capacitors are recommended to be placed on intended power pins. Decaps must be placed within 5 mm distance from the module pins.
5. The VINBCKDC capacitor should be placed within 10mm distance from the module pin.
6. Buck Inductor should be placed within 5 mm distance from the VOUTBCKDC pin. Buck capacitor should be placed within 1 mm distance from the Buck Inductor.
7. Series resistor on Clock signal of SPI or SDIO interface must be placed close to the source of this clock signal. It must not be placed near the module end of the signal.

Considering the above recommendations, the B00 module and its circuitry can be placed on a single side of PCB as shown below. This takes less than 25 mm x 15 mm of board space approximately, until RF front-end discrete components. The designer can choose to place the components different from the below setup to suit their product needs.

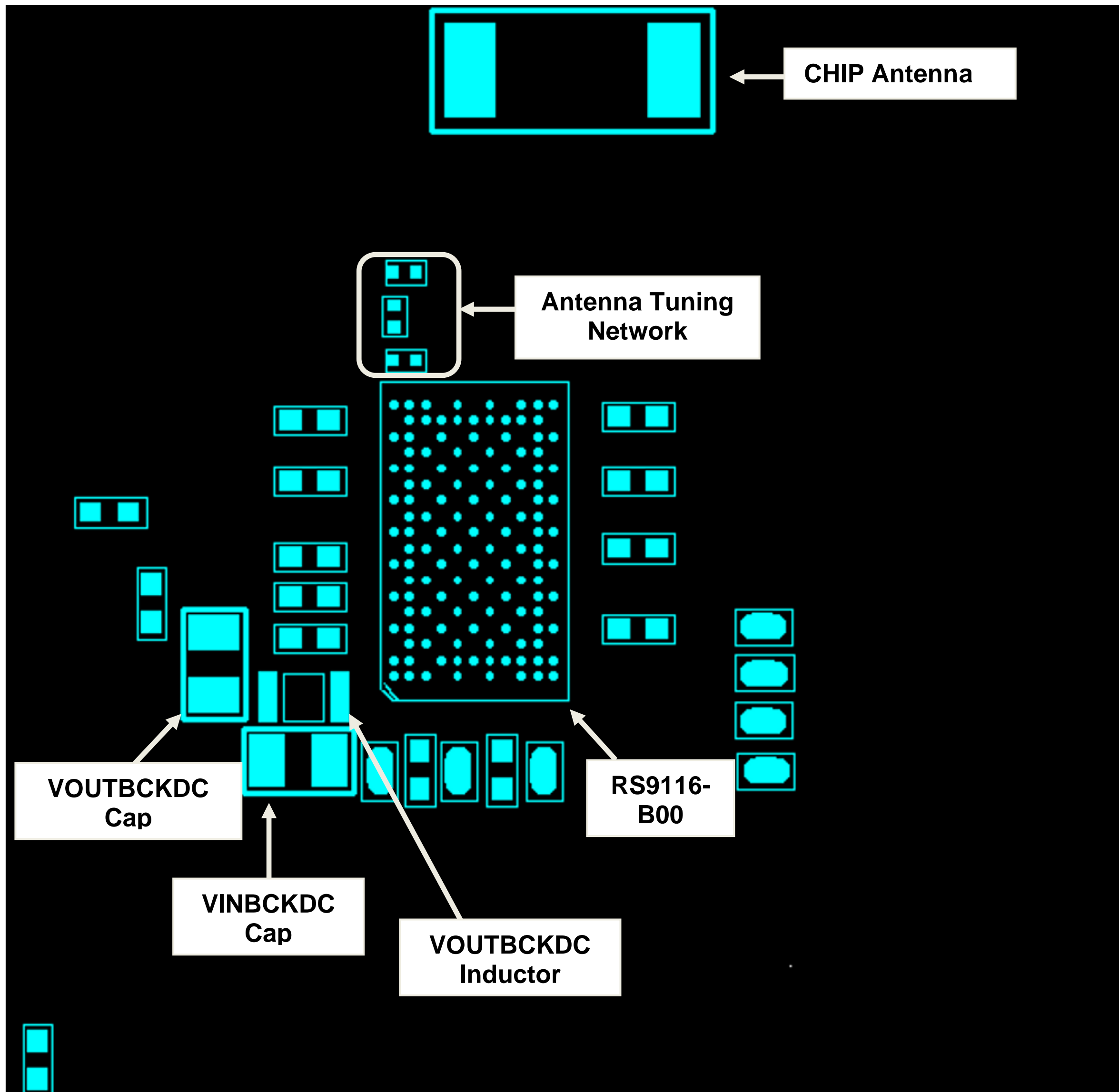

Figure 1: B00 Module and Components Placement

3 RF Layout Guidelines

RF parts placement and traces routing affect the RF performance of the module and circuitry. Great care must be taken to ensure best design practices and below guidelines are followed.

1. The RF trace on RF_PORT1 should have a characteristic impedance of 50 Ω. Any standard 50 Ω RF trace (Microstrip or Coplanar wave guide) may be used. The width of the 50 Ω line depends on the PCB stack, e.g., the dielectric of the PCB, dielectric constant of the material, thickness of the dielectric and other factors. Consult the PCB fabrication unit to get these factors right.
2. There must be DC blocking capacitors (8.2 pF) on RF_PORT1 if they are connected to Antenna
3. A thicker trace allows more manufacturing tolerance while keeping 50 Ω impedance.
4. Keep the length of the RF traces as short as possible.
5. Route RF traces on the top layer as much as possible and use a continuous reference ground plane underneath them.
6. Maintain at least one trace-width of space between nearby GND pour and the RF trace.
7. Add GND Stitches around RF traces.
8. Do not route any digital or analog signal traces between the RF traces and the reference ground.
9. Etch the GND copper underneath the antenna in all layers.
10. Follow the antenna placement as per the antenna vendor layout guidelines.
11. To evaluate transmit and receive performance like Tx Power and EVM, Rx sensitivity and the like, an RF connector would be required. A suggestion is to place a 'microwave coaxial connector with switch' between RF_OUT and the antenna.
12. The RF_PORT1 signal may be directly connected to an on-board chip antenna or terminated in an RF connector of any form factor for enabling the use of external antennas.
13. Use ground pour on the top and bottom layers in the RF area with plenty of ground stitch vias. Use stitching in a staggered pattern with a minimum via spacing of 32 mil (via to via, 25-mil offset) and a maximum spacing of 100 to 150 mil. After ground flooding, terminate shapes and corners with vias to tie to other planes for improved EMI performance.

Considering the above recommendations and the placement guidelines described in earlier section, possible RF circuitry routing in Layer 1 is shown below. Major parts and traces are also highlighted below.

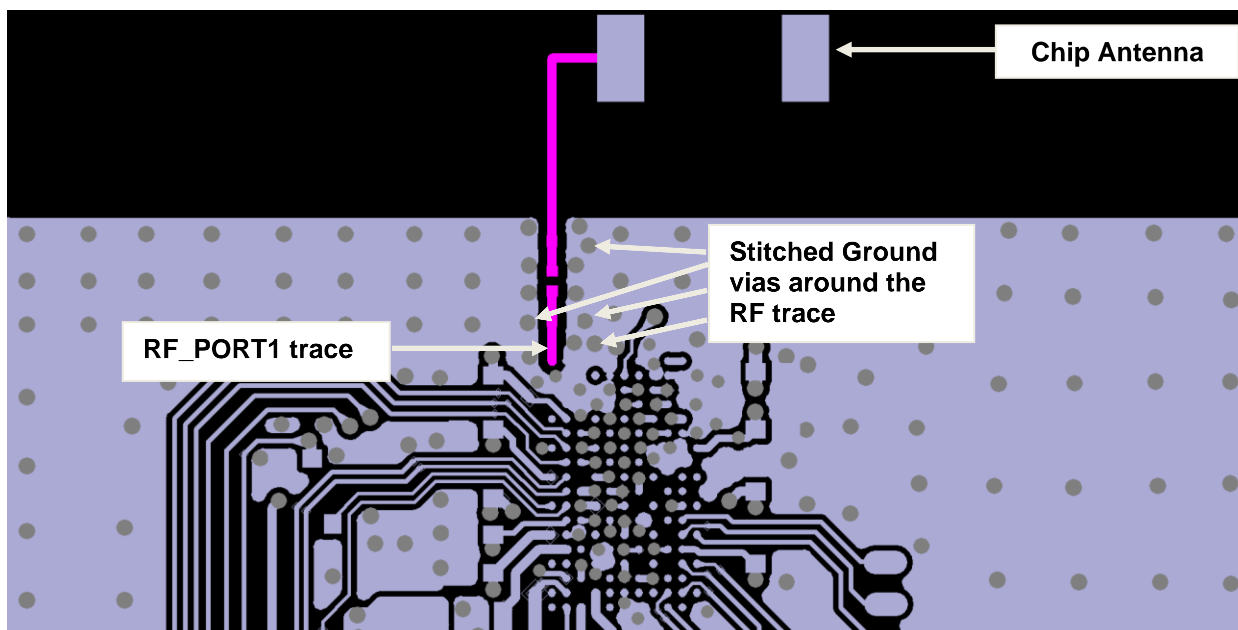

Figure 2: RF Circuitry Routing in Layer 1

The image below shows the overlap of RF routing on Top layer and the GND plane in the second layer. As can be observed, the entire RF circuitry has a second layer as a GND reference. GND vias around RF circuitry are stitched directly to the GND plane.

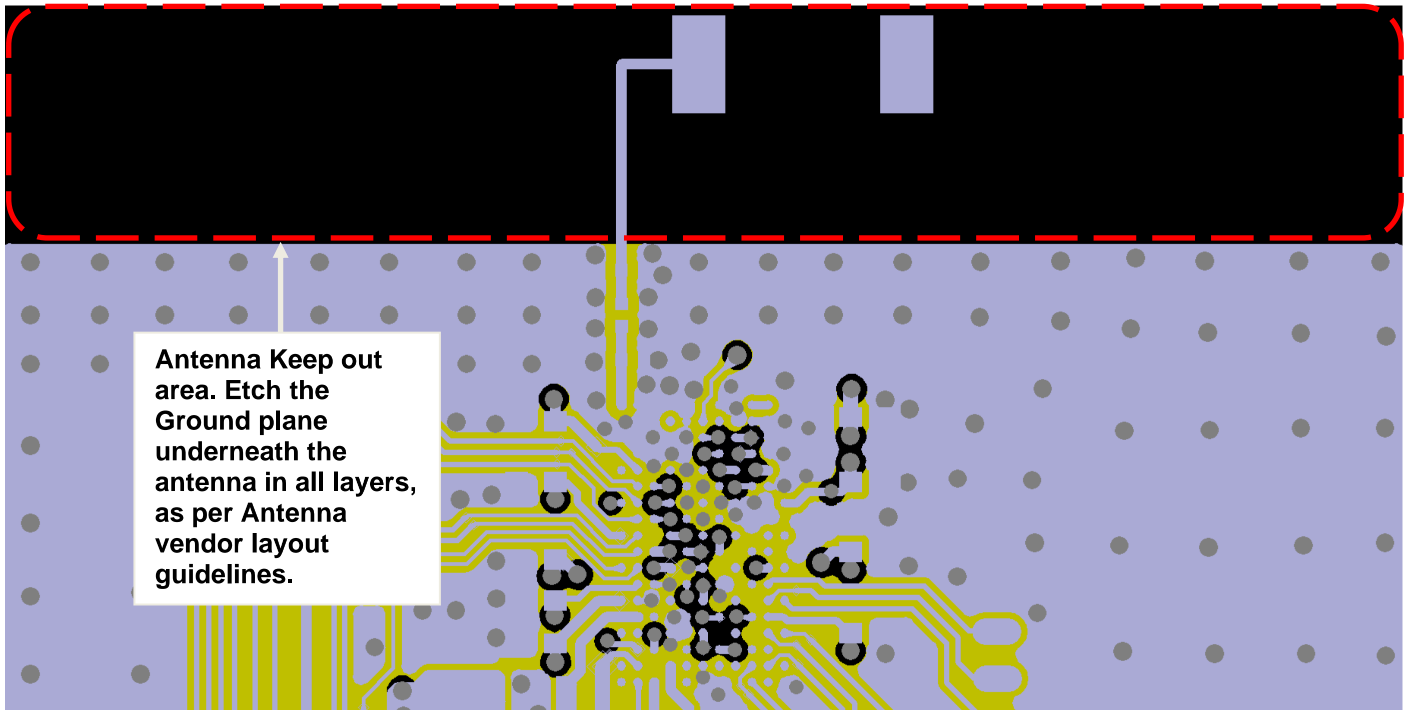

Figure 3: RF Circuitry Overlap with GND Layer

4 SDIO/SPI Layout Guidelines

SDIO and SPI are high-speed interfaces where-in clock signals can reach up to 100 MHz. The high-speed design guidelines like below must be followed for the signals in these interfaces. The SDIO/SPI signals include SDIO_CLK/SPI_CLK, SDIO_CMD/SPI_CSN, SDIO_D0/SPI_MOSI, SDIO_D1/SPI_MISO, SDIO_D2/SPI_INTR, SDIO_D3.

1. The characteristic impedance of the SDIO lines should be 50 Ω.
2. Match the lengths of all SDIO lines within 100 mils tolerance.
3. Keep the SDIO_CLK trace away from nearby traces with minimum 2x distance.
4. Do not route the parallel trace above or underneath the SDIO_CLK trace.
5. Keep SDIO traces away from all clock lines and noisy power supply components such as the switcher inductors. Avoid crossing-over power supplies or ground discontinuities. Ideally, the SDIO traces should have a solid ground on the layer adjacent to them.
6. Do not leave any stubs on the SDIO traces.

The Image below shows the SDIO/SPI traces routing in Layer 1, as an example. The series resistor on SDIO_CLK/SPI_CLK is placed closer to the source of this clock, rather than placing it closer to the B00 module.

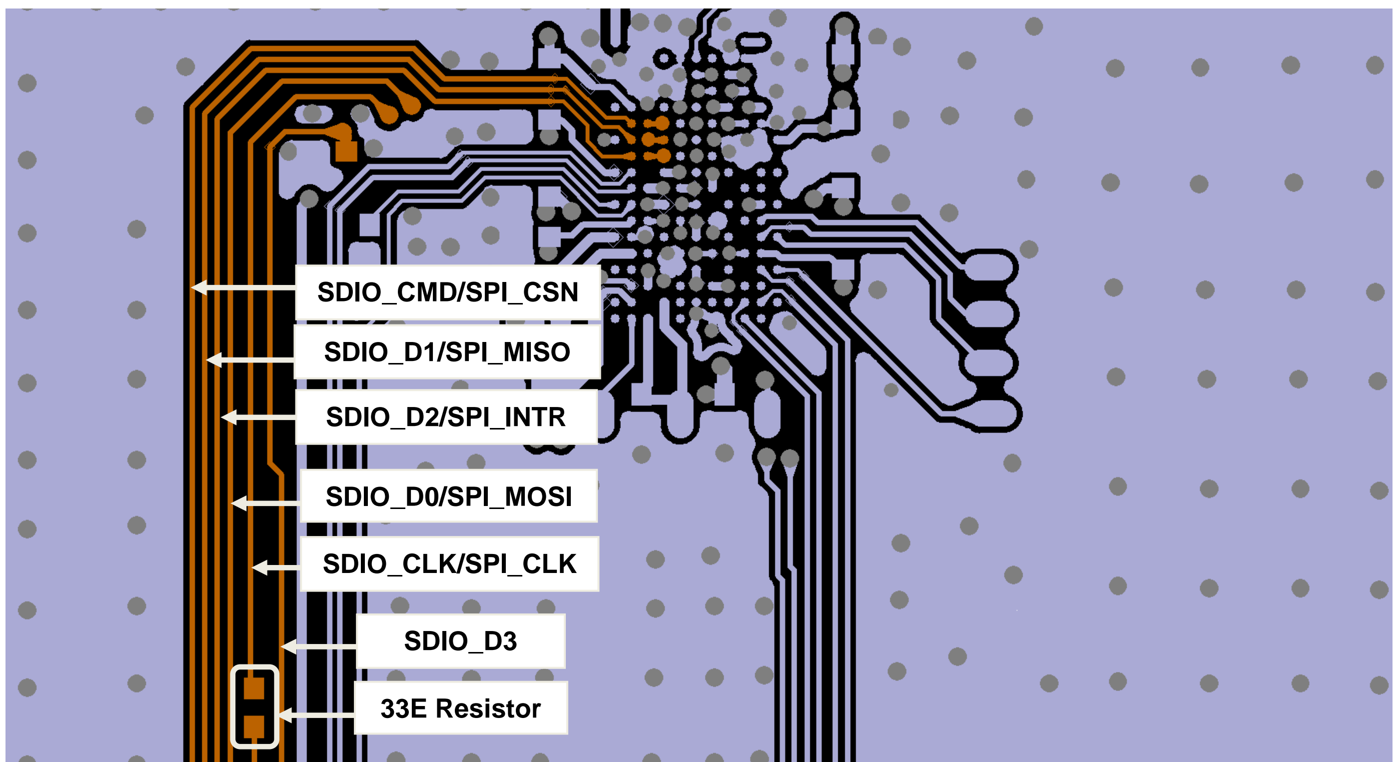

Figure 4: SDIO/SPI Signals Routing in Layer 1

The image below shows SDIO traces routing in Layer 4, as an example.

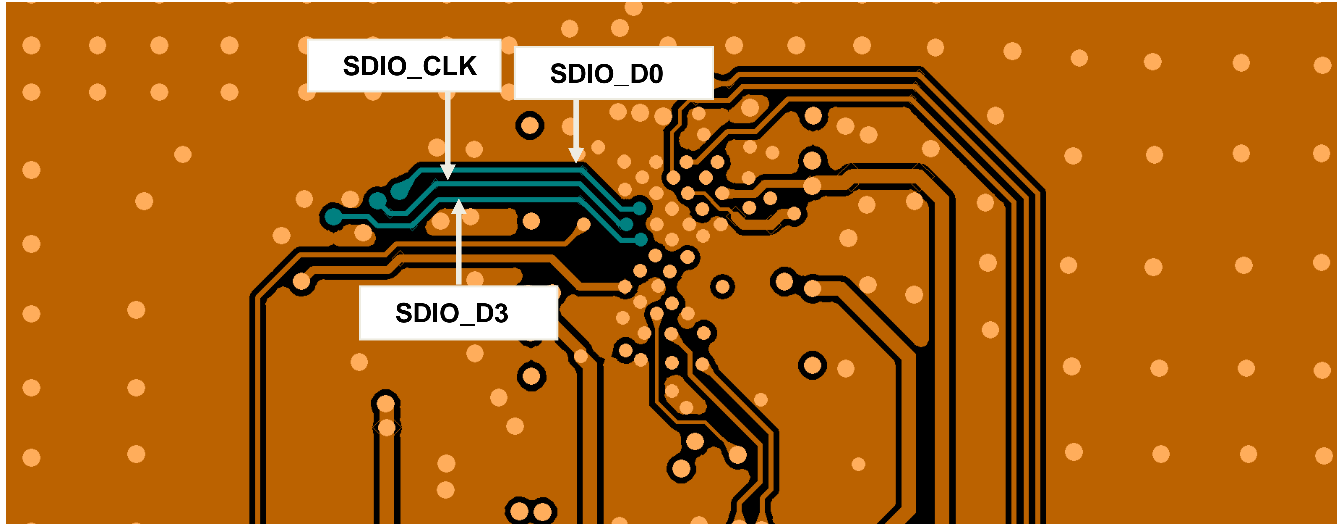

Figure 5: SDIO/SPI Signals Routing in Layer 4

5 USB Layout Guidelines

USB signals can reach speeds of 480 Mbps. Guidelines for the differential signals USB_DP and USB_DM must be followed.

1. It is highly recommended that the two USB differential signals (USB_DP and USB_DN) be routed in parallel with a spacing (say, a) that achieves 90 Ω of differential impedances and 45 Ω for each trace.
2. To minimize crosstalk between the two USB differential signals (USB_DP and USB_DN) and other signal traces routed close to them, it is recommended that a minimum spacing of 3xa be maintained for low-speed non-periodic signals and a minimum spacing of 7xa be maintained for high-speed periodic signals.
3. It is recommended that the total trace length of the signals between the RS9116 part and the USB connector (or USB host part) be less than 450mm.
4. If the USB high-speed signals are routed on the Top layer, best results will be achieved if Layer2 is a continuous GND plane. Furthermore, there must be only one ground plane under high-speed signals and avoid the high-speed signals crossing from GND plane to another GND plane.
5. Do not route the USB differential lines close to edge of the board.

The image below shows USB differential traces routing in Layer 1, as an example.

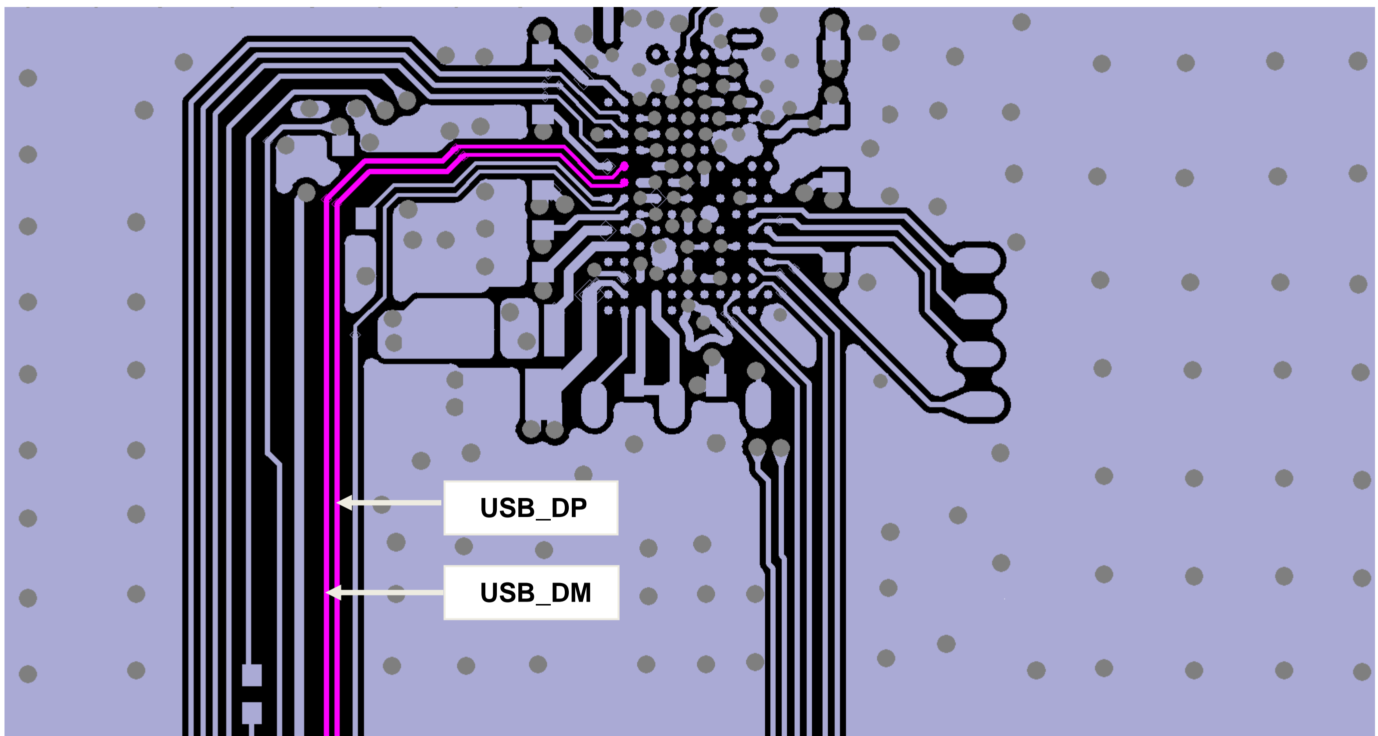

Figure 6: USB Signals Routing in Layer 1

6 UART Layout Guidelines

The guidelines below must be followed for UART signals. The signals are UART1_TX, UART1_RX, UART1_RTS, UART1_CTS.

1. Keep the UART signals away from noisy sources or other sensitive signals.
2. UART signals can be routed with multiple vias. However, ensure return path is closer to the signals.

The image below shows UART traces routing in Layer 4, as an example.

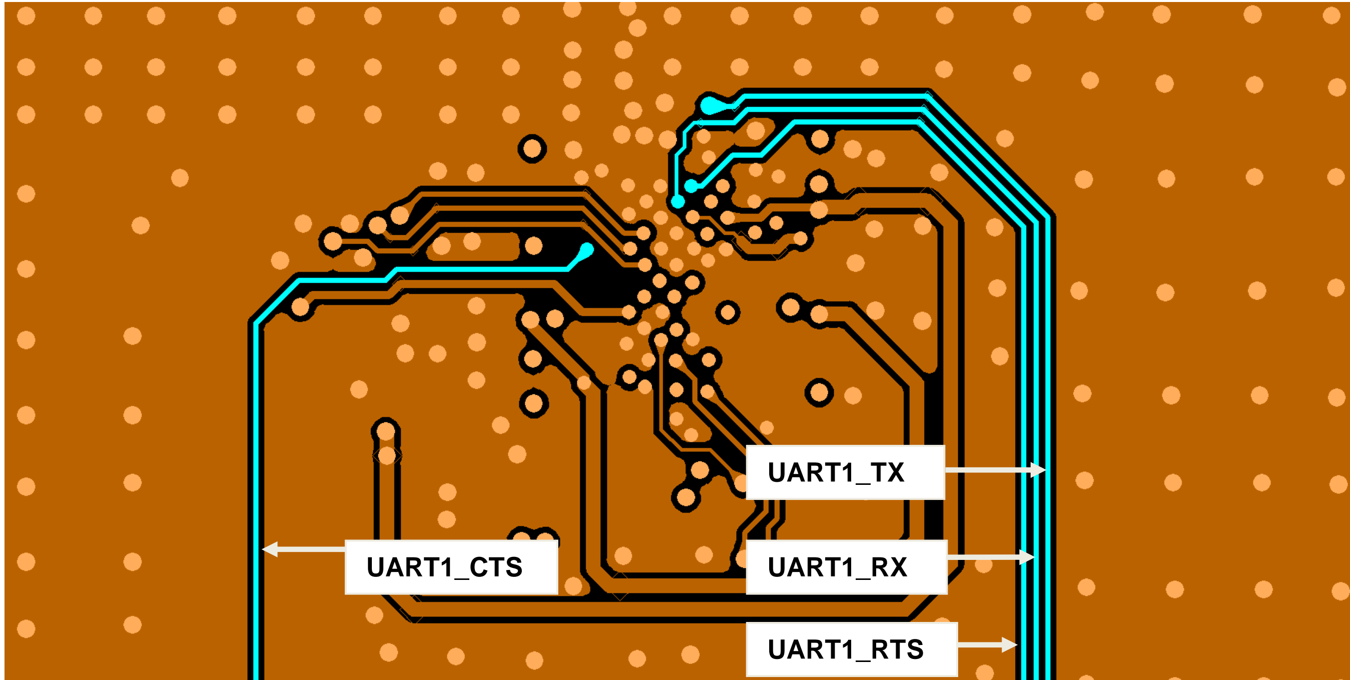

Figure 7: UART Signals Routing in Layer 4

7 Power Supply Layout Guidelines

There are many power pins on the B00 module. Careful routing with appropriate trace widths must be followed for better power delivery to the module. Follow the below guidelines for all the power traces.

1. The following power supply pins needs to be STAR routed from the Supply Source.
    a. VINBCKDC
    b. VINLDO1P8
    c. IO_VDD_1, IO_VDD_2
    d. ULP_IO_VDD
    e. UULP_VBATT_1
    f. UULP_VBATT_2
    g. RF_VBATT
    h. PA2G_AVDD
    i. SDIO_IO_VDD
2. The layout guidelines for the BUCK are as below.
   Minimize the loop area formed by inductor switching node, output capacitors and input capacitors. This helps keep high current paths as short as possible. Keeping high current paths shorter and wider would help decrease trace inductance and resistance. This would significantly help increase the efficiency in high current applications. This reduced loop area would also help in reducing the radiated EMI that may affect nearby components.
    a. VINBCKDC Capacitor should be very close to the Chip Pin and the Ground Pad of the capacitor should have direct vias to the Ground Plane underneath.
    b. Buck Inductor should be close to Chip VOUTBCKDC pin and buck capacitor should be placed closer to the Inductor. The Ground Pad of the capacitor should have direct vias to the Ground Plane underneath.
    c. The Ground Plane underneath the Buck Inductor in the Top Layer should be made as an isolated copper patch and should descend to the Second Layer (Main Ground) through multiple Vias.
    d. The path from VOUTBCKDC to VINLDOSOC is a high current path. The Trace should be as short and wide as possible and is recommended to run Grounded Shield Traces on either side of this High Current Trace.
    e. The Capacitor on VINLDOSOC should be very close to the Chip Pin, and the Ground Pad of the capacitor should have direct vias to the Ground Plane underneath.
3. If any power supply trace or shape needs to change layers from a layer referencing (adjacent to) the top ground plane to the bottom ground plane in the stack-up or vice versa, an equal number of ground vias should be interspersed with, or placed immediately adjacent to the vias carrying the supply voltage. This minimizes noise coupling from the supply to nearby signals and other supplies.
4. The width of power traces must be a minimum of 15 mils. Verify that they are wide enough to support target currents, and that they can do so with margin. Verify that there are enough vias if the traces need to change layers.
5. Place decoupling capacitors near target power pins. If possible, keep them on the same side as the module, to avoid inductance due to vias.

The image below shows example layout of Internal Buck regulator's Power Supply traces routing in Layer 1.

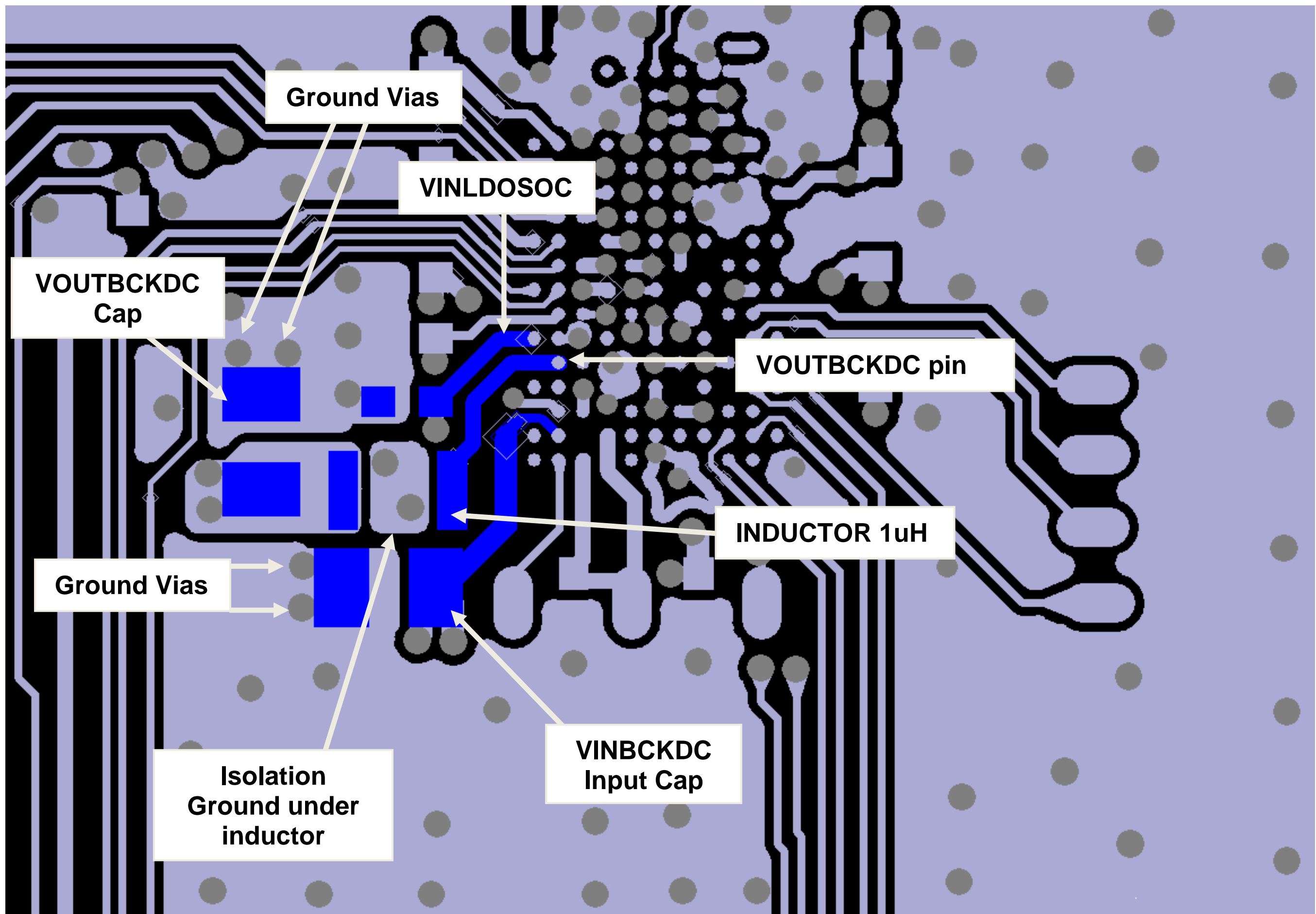

Figure 8: Internal Buck Regulator Circuitry Routing in Layer 1

The image below shows some of the Power Supply traces routing in the Layer 3. As shown, they are routed in Star fashion from the supply source.

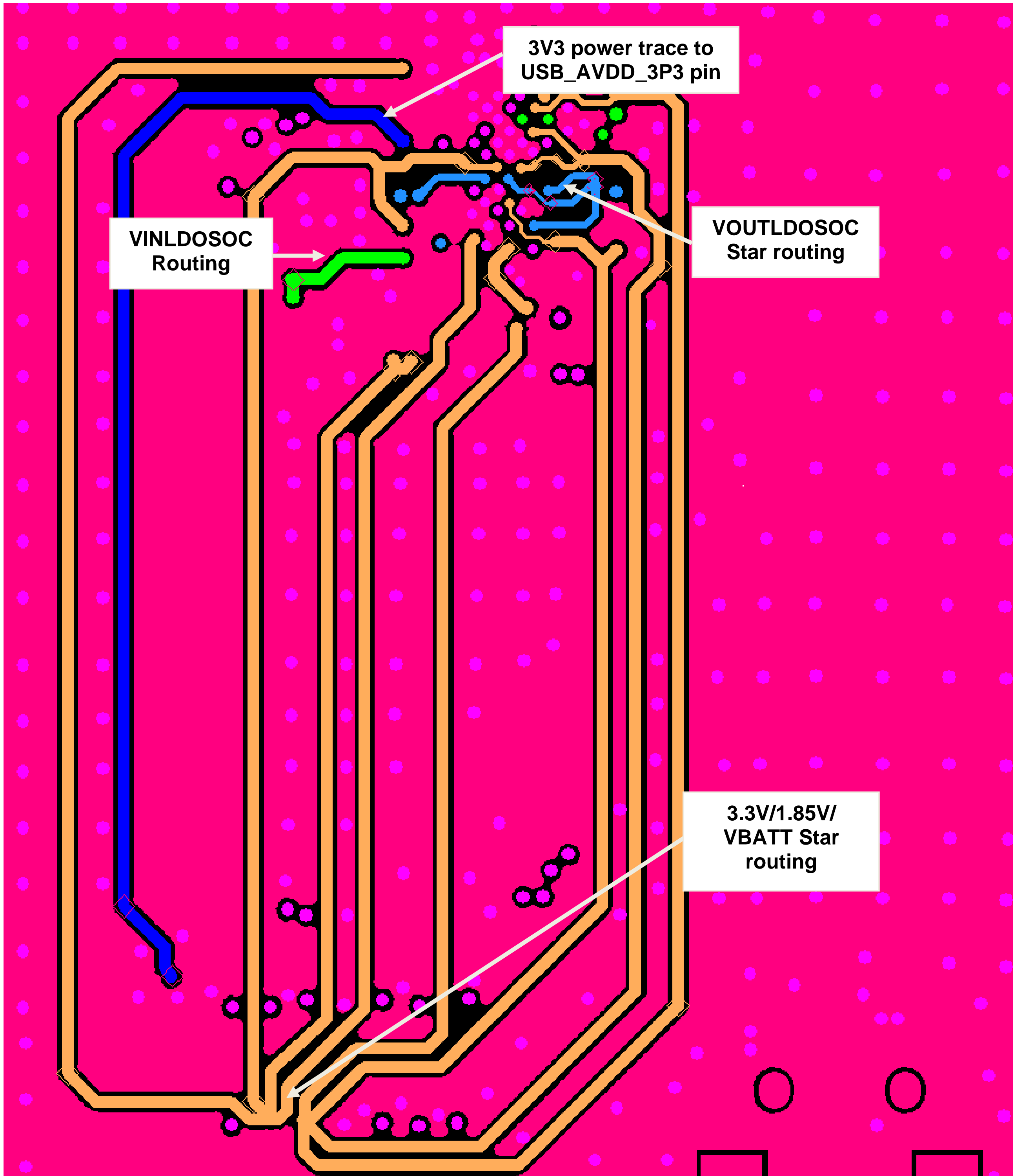

Figure 9: Power Supply Routing in Layer 3

The image below shows Power Supply traces routing in the Layer 4. As shown, they are routed in Star fashion from the supply source.

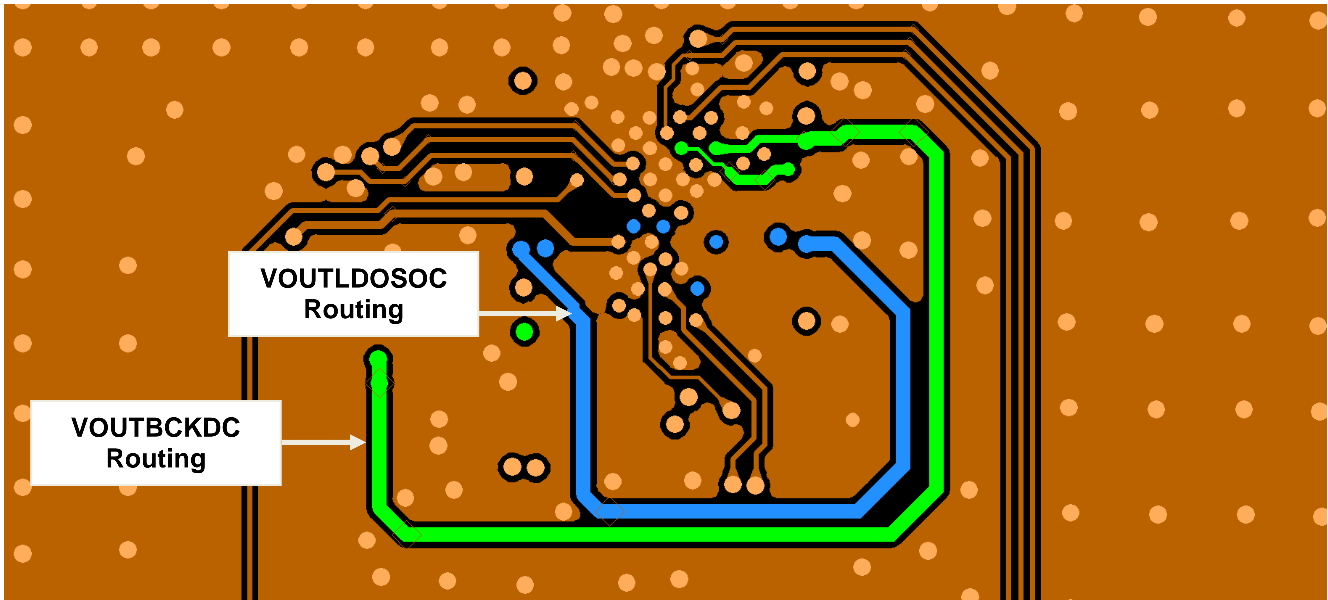

Figure 10: Power Supply Routing in Layer 4

8 GND Pour Layout Guidelines

All the returns paths of critical signals like RF, high-speed signals like SPI/SDIO/USB and Power, flow through GND. So, GND carries lot of return currents and high-speed signals. The designer must ensure return paths are short. There are various possible ways to achieve good grounding, and below guidelines provide some of the possible insights into it.

1. Dedicate the adjacent layer of the B00 module and its circuitry, to GND. In this application note, the B00 module and the circuitry are placed in Layer 1, so Layer 2 must be completely GND plane only.
2. GND pour must be continuous with no voids.
3. Pour GND in all the empty spaces around parts and traces in all the layers. Stitch this GND pour to GND plane using multiple GND vias.
4. Avoid large GND fill without GND vias or else it acts like an antenna, and this can possibly cause radiation of unwanted signals affecting other parts of the board. This can negatively affect board performance.

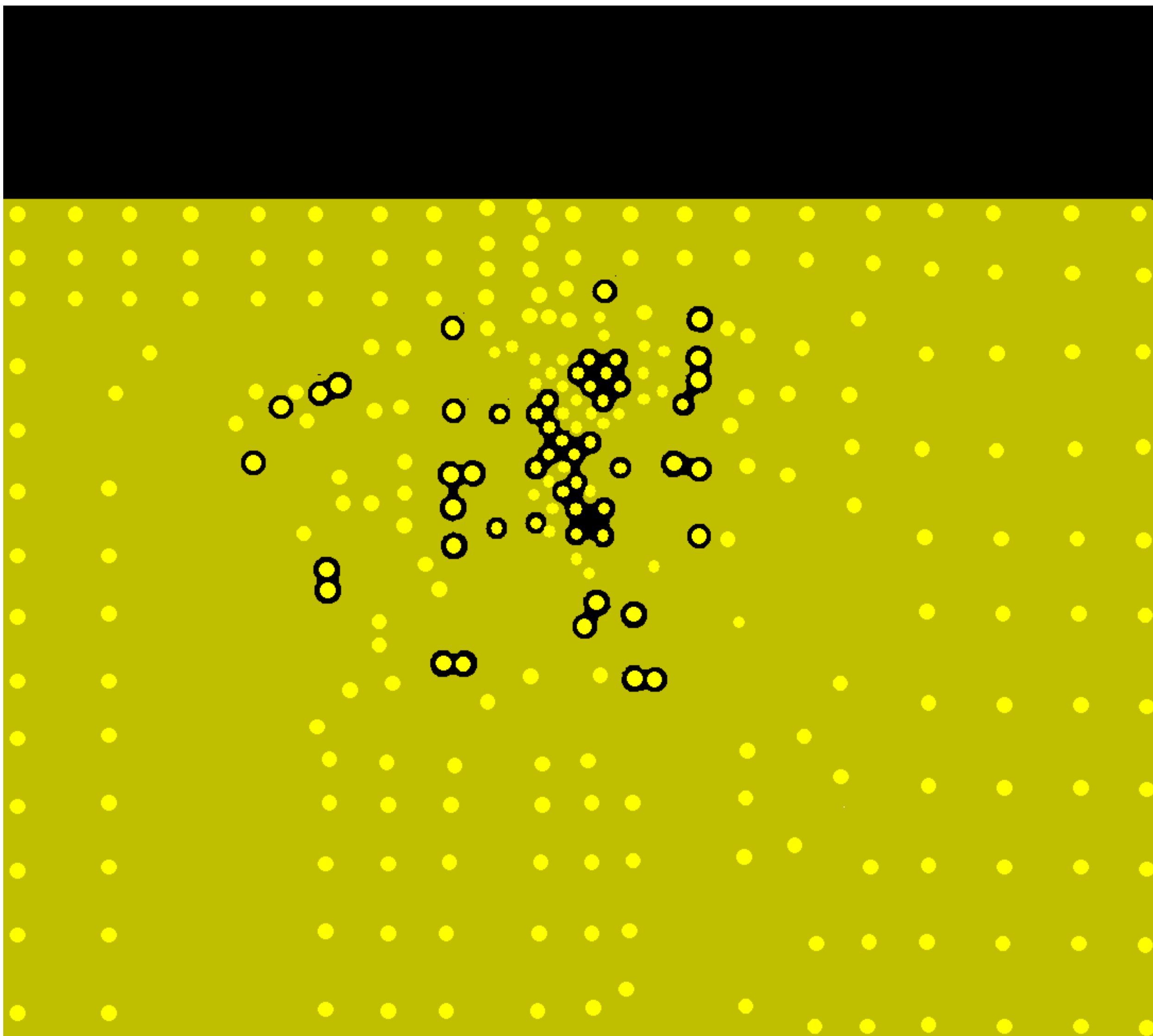

Figure 11: GND Plane in Layer 2

The image below shows the example picture of GND pour with one just via. Such hanging GND pour must be avoided. Stitch GND vias near the periphery of the GND pour, or else restrict such GND pour shapes.

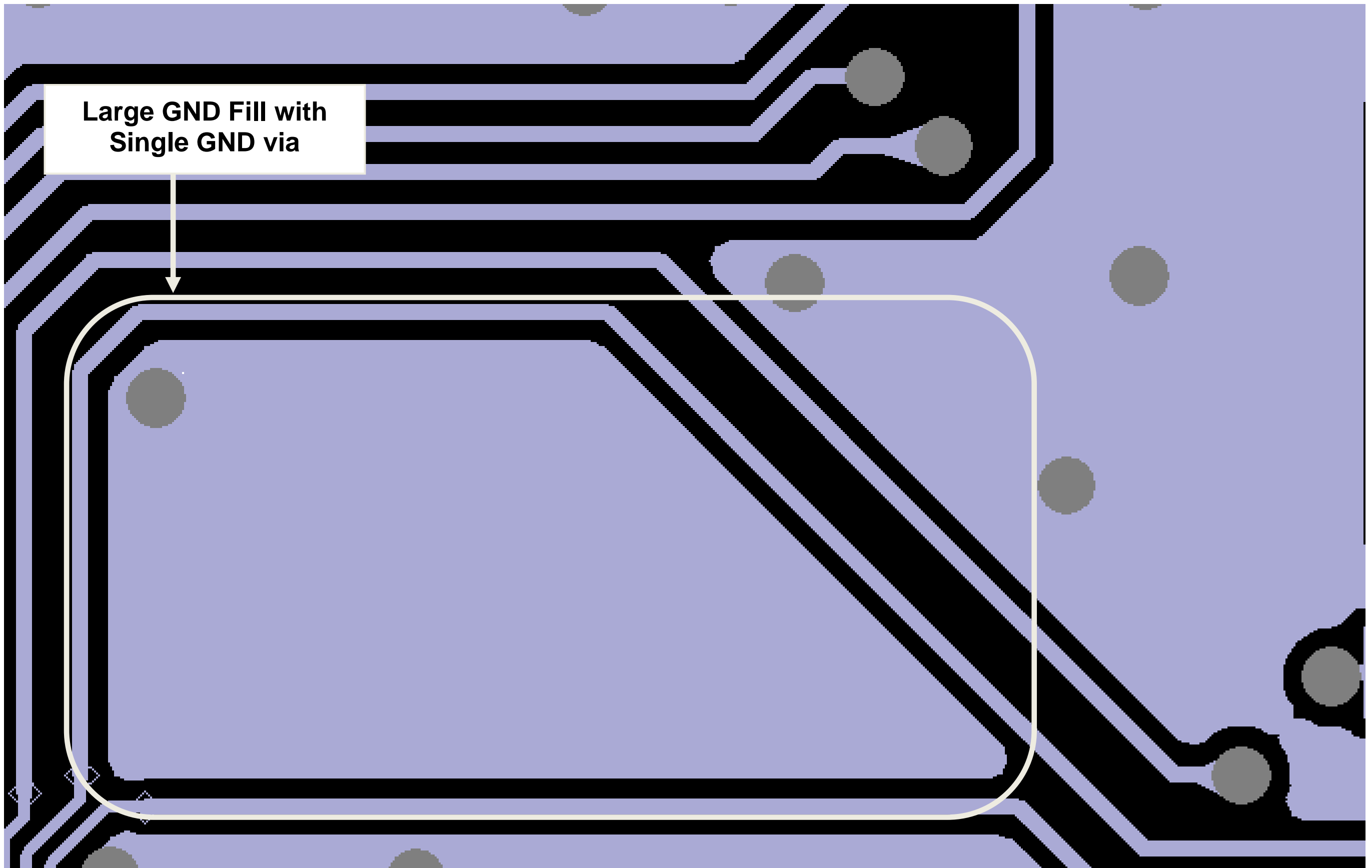

Figure 12: Hanging GND Pour

Simplicity Studio

One-click access to MCU and wireless tools, documentation, software, source code libraries & more. Available for Windows, Mac and Linux!

IoT Portfolio
www.silabs.com/IoT

SW/HW
www.silabs.com/simplicity

Quality
www.silabs.com/quality

Support & Community
www.silabs.com/community

Disclaimer
Silicon Labs intends to provide customers with the latest, accurate, and in-depth documentation of all peripherals and modules available for system and software implementers using or intending to use the Silicon Labs products. Characterization data, available modules and peripherals, memory sizes and memory addresses refer to each specific device, and "Typical" parameters provided can and do vary in different applications. Application examples described herein are for illustrative purposes only. Silicon Labs reserves the right to make changes without further notice to the product information, specifications, and descriptions herein, and does not give warranties as to the accuracy or completeness of the included information. Without prior notification, Silicon Labs may update product firmware during the manufacturing process for security or reliability reasons. Such changes will not alter the specifications or the performance of the product. Silicon Labs shall have no liability for the consequences of use of the information supplied in this document. This document does not imply or expressly grant any license to design or fabricate any integrated circuits. The products are not designed or authorized to be used within any FDA Class III devices, applications for which FDA premarket approval is required or Life Support Systems without the specific written consent of Silicon Labs. A "Life Support System" is any product or system intended to support or sustain life and/or health, which, if it fails, can be reasonably expected to result in significant personal injury or death. Silicon Labs products are not designed or authorized for military applications. Silicon Labs products shall under no circumstances be used in weapons of mass destruction including (but not limited to) nuclear, biological or chemical weapons, or missiles capable of delivering such weapons. Silicon Labs disclaims all express and implied warranties and shall not be responsible or liable for any injuries or damages related to use of a Silicon Labs product in such unauthorized applications. **Note: This content may contain offensive terminology that is now obsolete. Silicon Labs is replacing these terms with inclusive language wherever possible. For more information, visit www.silabs.com/about-us/inclusive-lexicon-project**

Trademark Information
Silicon Laboratories Inc.®, Silicon Laboratories®, Silicon Labs®, SiLabs® and the Silicon Labs logo®, Bluegiga®, Bluegiga Logo®, EFM®, EFM32®, EFR, Ember®, Energy Micro, Energy Micro logo and combinations thereof, "the world's most energy friendly microcontrollers", Redpine Signals®, WiSeConnect , n-Link, ThreadArch®, EZLink®, EZRadio®, EZRadioPRO®, Gecko®, Gecko OS, Gecko OS Studio, Precision32®, Simplicity Studio®, Telegesis, the Telegesis Logo®, USBXpress® , Zentri, the Zentri logo and Zentri DMS, Z-Wave®, and others are trademarks or registered trademarks of Silicon Labs. ARM, CORTEX, Cortex-M3 and THUMB are trademarks or registered trademarks of ARM Holdings. Keil is a registered trademark of ARM Limited. Wi-Fi is a registered trademark of the Wi-Fi Alliance. All other products or brand names mentioned herein are trademarks of their respective holders.

Silicon Laboratories Inc.
400 West Cesar Chavez
Austin, TX 78701
USA

www.silabs.com