ENVISIONING • EMPOWERING • EXCELLING

RS-232 and RS-485 PCB Layout

Application Note

Revision History

Document No.	Release Date	Change Description
293ANR00	December 1, 2022	Initial release.

Table of Contents

List of Figures

Introduction

This document describes recommended printed circuit board (PCB) design and layout practices for RS-232 and RS-485/RS-422 serial transceiver products.

The PCB is one of the most important factors that affect device performance, ESD and EMI. The expected PCB design minimizes the system noise and provide shielding between the PCB's internal circuitry and the external environment. MaxLinear recommends that the PCB layout practices described in this application note apply to the serial products and the entire system PCB design.

General Recommendation

This section list the tips you can follow for your PCB design.

Tip #1: Use power supply decoupling capacitors and place them close to the device supply pin

Voltage sources in electronic systems can generate a lot of noise that affects both digital and analog circuits. Analog systems are extremely prone to supply voltage noise. This unwanted noise can distort the analog signal resulting in false values detected by devices further down the signal path.

In most systems, the direct current (DC) supply is a constant voltage with ripple less than 50mV. Although small, it is good practice to minimize these supply voltage noises by adding decoupling capacitors to the device supply pin.

- Connect a low-ESR, 0.1μF (or 0.01μF) ceramic bypass capacitors between each supply pin and ground.
- Always keep the filtering bypass capacitors as close to the device supply pin as possible.

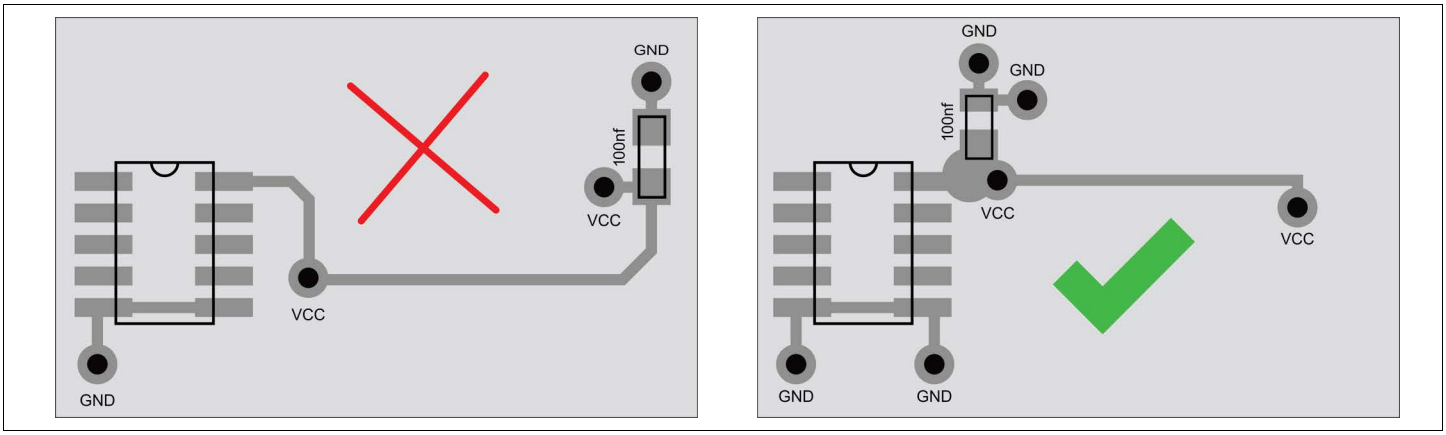

Figure 1: Optimal Power Supply Decoupling Capacitor Location

- Keep traces as short as possible to reduce the trace inductance.
- Use a solid ground plane and solid power plane instead of ground traces and power traces. This lowers the current-path inductance and ensures the expected supply to each component in the system.

- Place solid or filled vias into solid ground planes or power planes. Vias provide excellent connections and lower thermal dissipation.

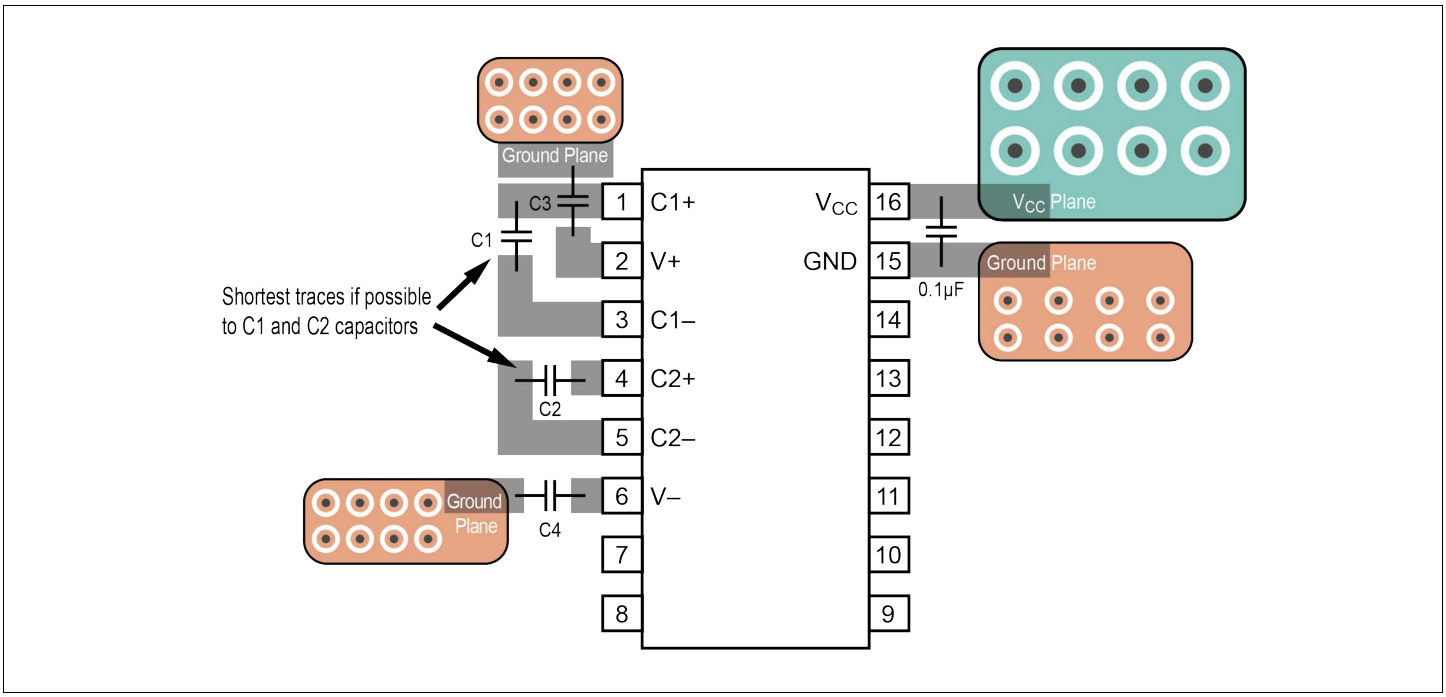

Figure 2: Optimal Use of Power Supply Traces, Ground Planes, and Power Planes

Tip #2: Keep Input and output traces short and tie unused pins to ground

- To reduce the noise coupling to input traces, run the input traces as far away from output traces and power supply sources.
- Always keep the length of the input traces as short as possible because the input traces are the most sensitive part of the device.
- Unused input pins must be tied to ground via 50Ω or directly to ground.

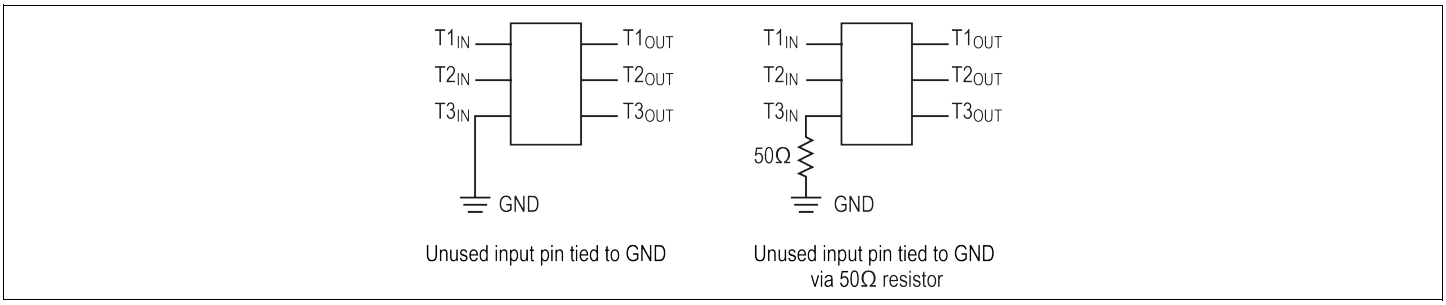

Figure 3: Input Traces and Unused Input Pin Strapping Recommendation

- Avoid routing traces at right angles to prevent from generating radiations and reflections.

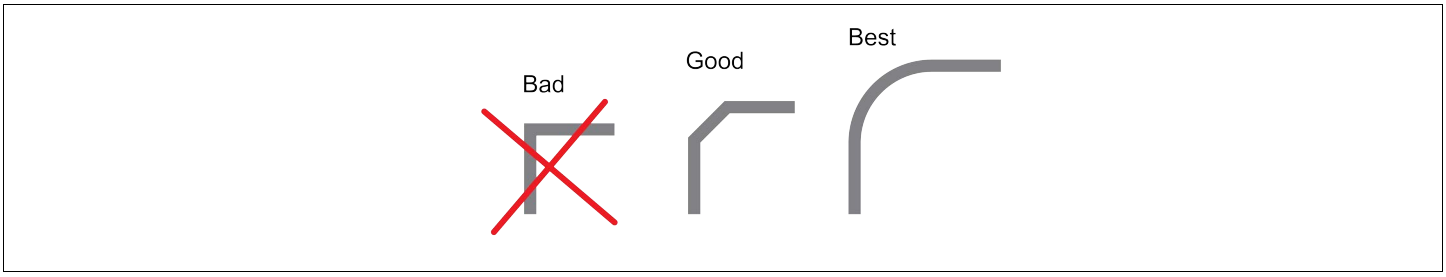

Figure 4: Trace Routing Recommendation

Tip #3: Digital switching signals and clock lines

Digital signals are binary with a defined voltage level. These signals transition between specific high and low voltage levels. They are used to convey logic data of the implemented protocols. Due to their sharp edge transitions and switching characteristics digital signals and clock lines can be a source of noise in the system layout. For critical applications, consider a low-impedance guard ring around critical traces. A guard ring can significantly reduce switching noise from coupling to nearby traces.

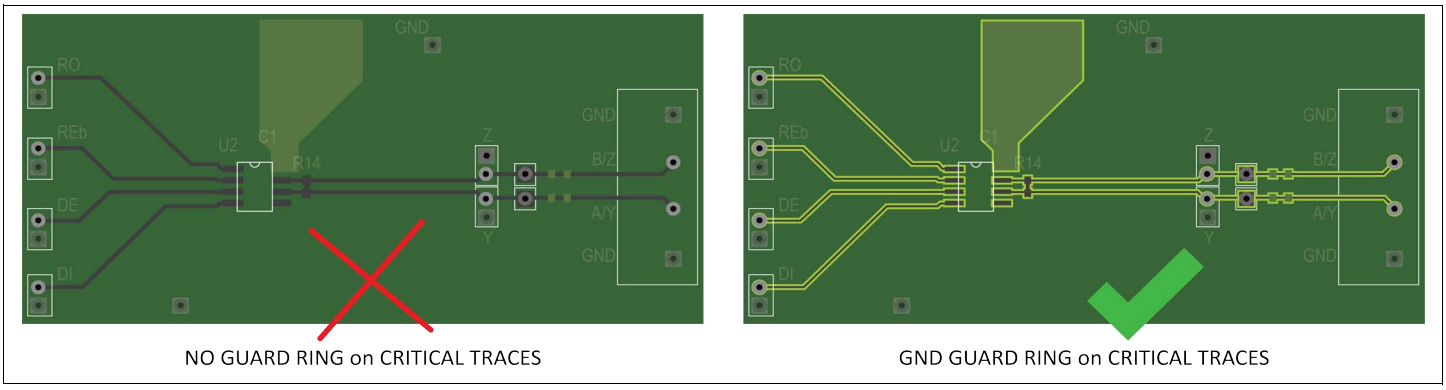

Figure 5: Example of a Guard Ring around Critical Traces

Tip #4: General recommendations

- Place components on the PCB in such a way as to avoid long loop traces.

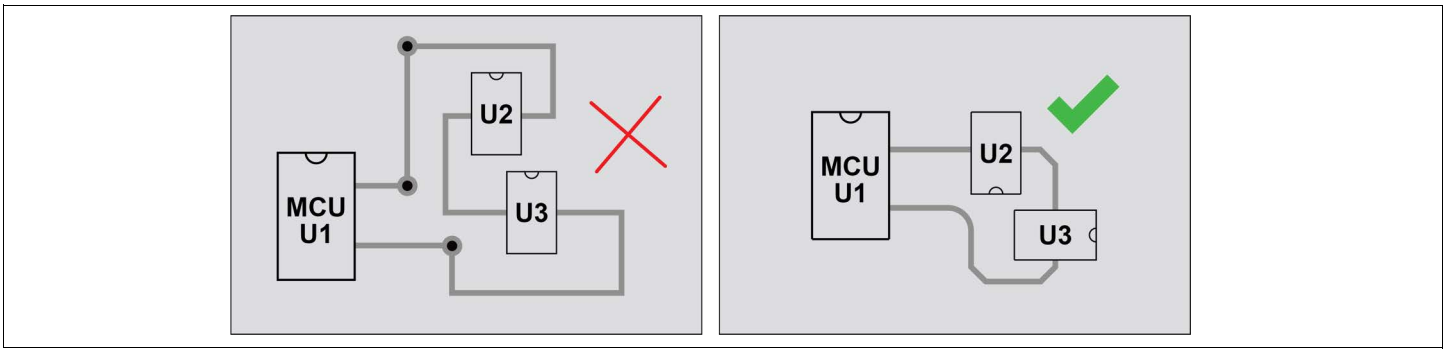

Figure 6: Component with Long Loop Traces Bad vs Good

- Avoid placing vias and pads in the path of critical signals.
  Vias and pads add additional unwanted capacitance and inductance which can cause signal reflection and distortion.
- Use a power supply with adequate voltage and current for the application and with ground noise less than 50mV peak-to-peak.
- Terminate all high-speed digital signal traces and clock lines to prevent reflections and signal ringing.

For additional support, contact your MaxLinear Contact Support team or visit www.maxlinear.com.

MAXLINEAR

Corporate Headquarters:
5966 La Place Court, Suite 100
Carlsbad, CA 92008
Tel.: +1 (760) 692-0711
Fax: +1 (760) 444-8598

www.maxlinear.com