



PL2731

USB 3.0 Single-LUN Mass Storage Card Reader

PCB Layout Guide

Document number : AN-27220901

Revision : 1.0

Release Date : September 27, 2022

Prolific Technology Inc.

7F, No. 48, Sec. 3, Nan Kang Rd.

Nan Kang, Taipei 115, Taiwan, R.O.C.

Telephone: +886-2-2654-6363

Fax: +886-2-2654-6161

E-mail: sales@prolific.com.tw

Website: <http://www.prolific.com.tw>

Table of Contents

Overview3

1. USB3.0 trace.....3

2. USB2.0 trace.....5

3. Power5

4. Others5

Overview

The guidelines provide the recommended consideration on PCB layout.

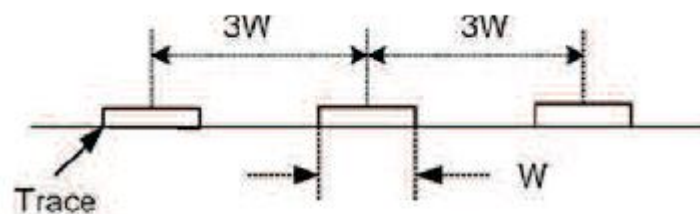
1. USB3.0 trace

- USB3.0 signal traces consists of two differential pair traces, a transmit pair (SSTX+ and SSTX-) and a receive pair (SSRX+ and SSRX-). Each differential pair traces should be routed with differential impedance of 90 ± 7 ohms.

- Never route USB3.0 differential traces near other high frequency traces such as the signal traces of USB2.0 and SD interfaces, and crystal to avoid crosstalk.

Ways to Avoid Crosstalk:

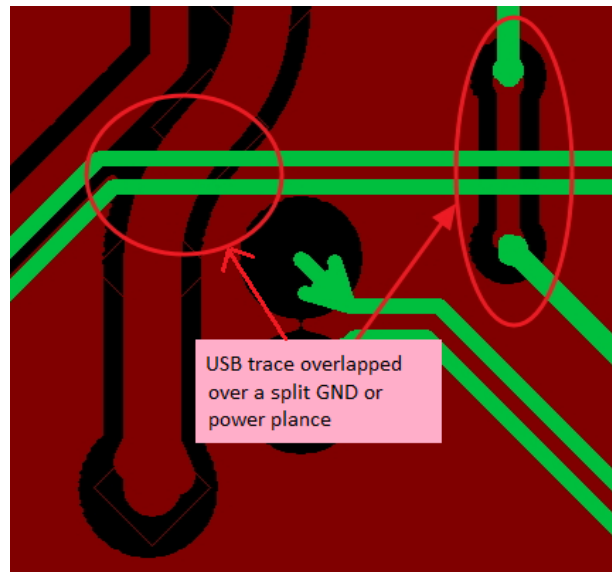
- Use the 3W rule (3 times the width of USB3.0 signal trace as figure 1) to separate USB3.0 signal trace from other high frequency traces.
- Use ground traces/guards around either USB3.0 signal or other high frequency signal traces.



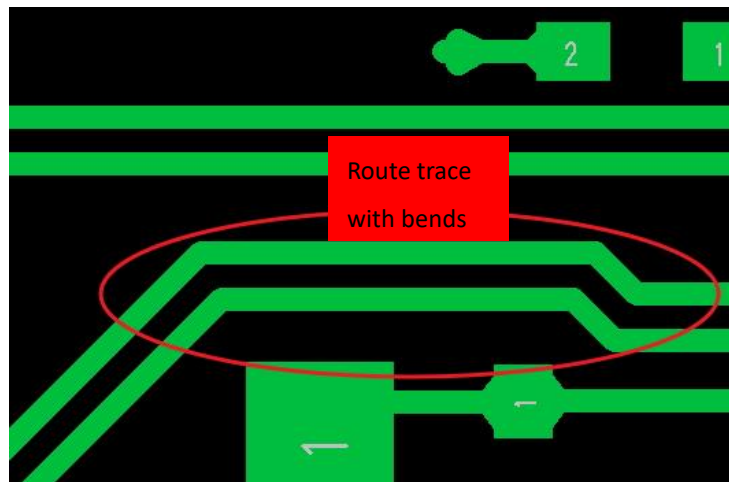
The "3W" rule

Figure 1

- Never route USB3.0 differential traces near the PCB edge to prevent from the distortion of USB3.0 signal by EMI and ESD.
- Never route USB3.0 differential traces over the split ground and power planes on the different and adjacent layer of PCB as figure 2.
For a worse example as Figure 2, USB3.0 differential pair is routed on the top layer of PCB and overlapped a split ground plane with other signal traces on the second layer(for 4-layer PCB) or the bottom layer(for 2-layer PCB) of PCB.

**Figure 2**

- It is allowed to swap the plus and minus traces of USB3.0 signal SSTX and SSRX. This can prevent one (ex. SSTX+) of the differential traces from crossing another differential trace(ex. SSTX-). However, it is not permissible to swap the transmitter differential pair (SSTX signal) with the receiver differential pair (SSRX signal).
- The length of one trace of each differential pair should be as the same as possible with another trace. Maximum trace length mismatch between USB3.0 signal pairs should be no greater than 100 mils.
- When using a common mode choke and an ESD protect part to suppress EMI and ESD, they should be placed as close as possible to the USB3.0 connector.
- The transmitter differential pair requires two 0.1 μ F coupling capacitors for proper operation. The package size of these capacitors should be no bigger than 0603 and placed as close as possible to signal pins of USB3.0 connector.
- It is recommended to straight route USB3.0 differential pair and less route them with bends as figure 3.
- Less routing of USB3.0 differential traces on the different layer of PCB. Routing them on the same layer without using via of PCB to change the layer is better. If routing them on the different layer cannot be avoided, first route the trace of the transmitter differential pair (SSTX signal) using via of PCB.

**Figure 3**

2. USB2.0 trace

- USB2.0 differential pair traces should be routed with differential impedance of $90 \pm 15\%$ ohms.
- The trace length of one trace of USB2.0 differential pair should be as the same as possible with another trace. Maximum trace length mismatch between USB signal pairs should be no greater than 100 mils.
- When using a common mode choke and an ESD protect part to suppress EMI and ESD, they should be placed as close as possible to the USB connector.
- Never route USB2.0 differential traces over the split ground and power planes on the different and adjacent layer of PCB.

3. Power

- Place decoupling capacitors near the power pin of PL2731.
- Minimum trace width of VBUS of USB connector is 20 mils.
- Minimum trace width of 1.2V, 3.3V and SD card power is 13 mils.

4. Others

- NC pins of 4-pin crystal should be connected to GND to prevent from generating EMI.
- The SD signal traces should be length matched. Maximum trace length mismatch should be no greater than 100 mils.

- Never route SD signal traces near other high frequency traces such as the signal traces of USB and crystal to avoid crosstalk.

Ways to Avoid Crosstalk:

- Use the 3W rule (3 times the width of SD signal trace) to separate SD signal trace from other high frequency traces.
- Use ground traces/guards around either SD signal or other high frequency signal traces.

Disclaimer

All the information in this document is subject to change without prior notice. Prolific Technology Inc. does not make any representations or any warranties (implied or otherwise) regarding the accuracy and completeness of this document and shall in no event be liable for any loss of profit or any other commercial damage, including but not limited to special, incidental, consequential, or other damages.

Trademarks

The Prolific logo is a registered trademark of Prolific Technology Inc. All brand names and product names used in this document are trademarks or registered trademarks of their respective holders.

Copyrights

Copyright © 2022 Prolific Technology Inc. All rights reserved.

No part of this document may be reproduced or transmitted in any form by any means without the express written permission of Prolific Technology Inc.

Prolific Technology Inc.

7F, No. 48, Sec. 3, Nan Kang Rd.
Nan Kang, Taipei 115, Taiwan, R.O.C.
Telephone: +886-2-2654-6363
Fax: +886-2-2654-6161
E-mail: sales@prolific.com.tw
Website: <http://www.prolific.com.tw>