

System Design Guidelines:

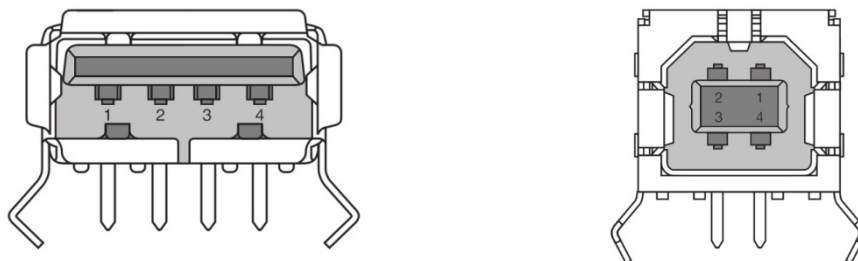
PCB Layout Guidelines for the USB 2.0

1 Introduction

This document provides guidelines for PCB design and circuit optimization of the USB portion of the ADChips solution platform.

USB Connector

A standard USB connector, either type A or type B, provides good ESD protection. However, if a custom USB connector is desired, the following guidelines should be observed to ensure good ESD protection.



PIN	Signal	Description
1	VCC	+5V
2	D- / DM	Inverted Data Line
3	D+ / DP	Data Line
4	GND	Ground

Figure 1-1 : USB Receptacle Type-A / Type-B

2 USB physical interface schematics

Place the USB Connector as close as possible the adStar USB block. The signal swing during high speed operation on the DP/DM lines is relatively small, so any differential noise picked up on the twisted pair can affect the received signal.

Connect the shield connection to GND through parallel a resistor and a capacitor. Keep this two components close to the USB connector.

An electrical schematic of the USB area and the component locations relative to each other, illustrated below:

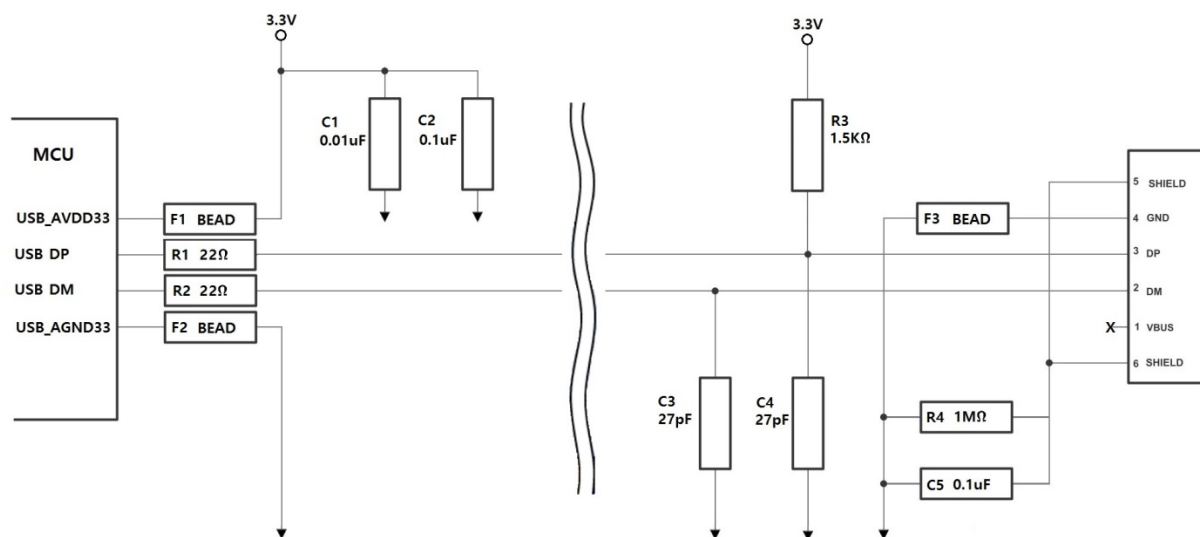


Figure 2-1: Trace Spacing

C1, C2 : Decoupling capacitor for the USB PHY Power.

R1, R2 : Series termination resistors.

F1, F2, F3 : For cleaning the power line.

R4, C5 : Chassis ground for the USB connector.

C3, C4 : Bypass capacitor is noise reduce for the USB DP/DM signals.

3 USB physical interface PCB layout

There should be an easily accessible shield plate next to the connector for air-discharge mode purpose.

Tie the outer shield of the connector to GND. When a cable is inserted into the connector, the shield of the cable should first make contact with the outer shield.

If the connector includes power and GND, the lead of power and GND need to be longer than the leads of signal.

The connector needs to have a key to ensure proper insertion of the cable.

Short the cable connector sockets directly to small chassis ground plane that exists immediately underneath the USB connector.

Place the USB Connector as close as possible the adStar USB block. The signal swing during high speed operation on the DP/DM lines is relatively small, so any differential noise picked up on the twisted pair can affect the received signal.

Connect the shield connection to GND through parallel a resistor and a capacitor. Keep this two components close to the USB connector.

DM and DP

Route DP/DM traces close together for noise rejection on differential signals, parallel to each other and within two mils in length of each other.

The USB data lines, DP and DM, should be routed as a differential pair.

Route differential signal pairs on the same layer.

The DP/DM traces widths and space width 8~10mils.

Keep away from the edge of the PCB.

Route DP/DM signals far from the clock line to reduce crystal jitter.

DP/DM traces should always be matched lengths and must be no more than 5 inches in length.



Figure 3-1: Trace Spacing

Avoid stubs when adding components to D+/D- signals. Where termination or bias resistors are needed, one terminal should be located directly on the trace.

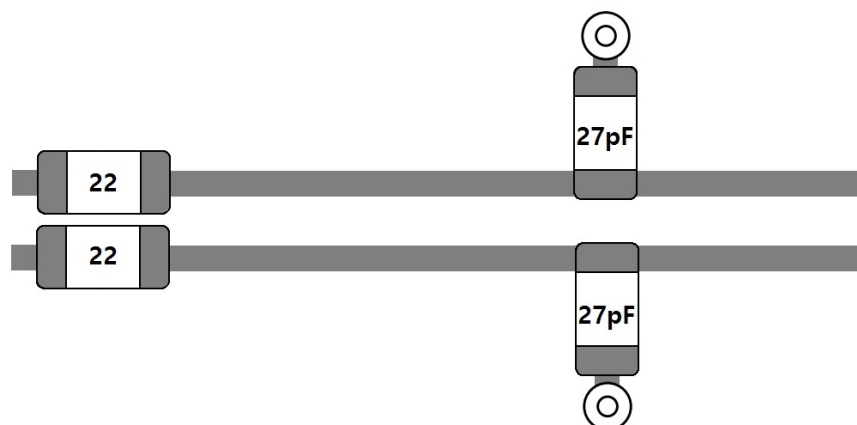


Figure 3-2: Avoid Stubs

Vias

Avoid routing DP and DM through vias.

Where vias are necessary keep them small (25-mil pad, 10-mil hole)

DP and DM signals in the pair should pass through a via at the same distance on the trace.

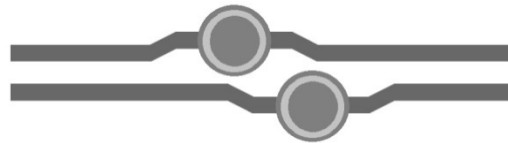


Figure 3-3: Vias

Ground

There must be only one ground plane under the USB signals in order to keep the USB signal from crossing another ground plane

Route DP and DM on the signal layer adjacent to the ground plane.

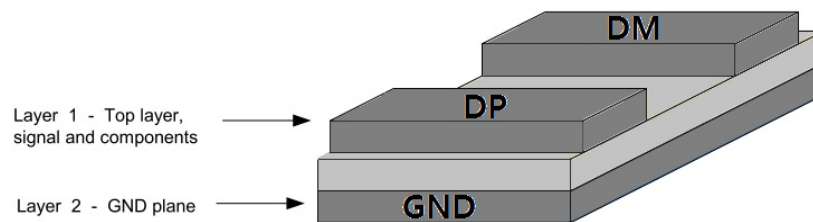


Figure 3-4: Ground Plane

4 Examples

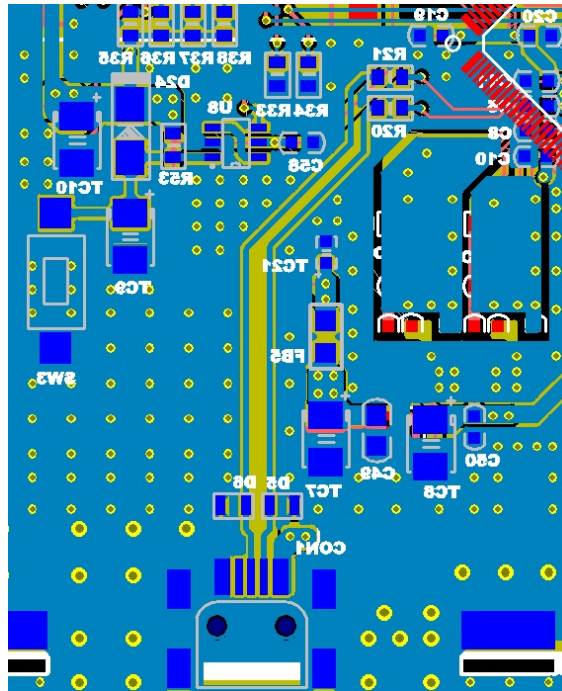


Figure 4-1: adStar

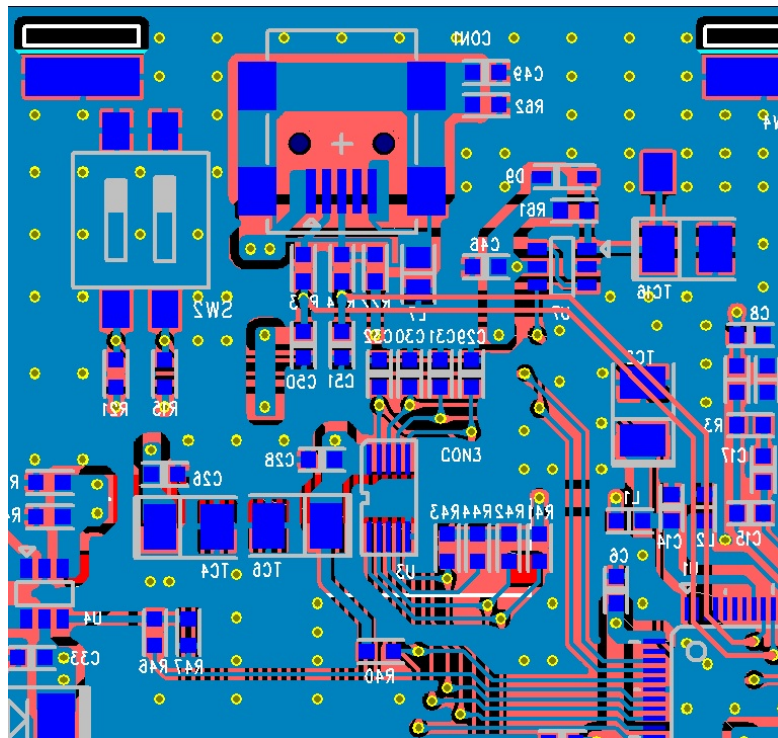


Figure 4-2: adStar-L