USB signal PCB routing guidelines:

Use the following guidelines to help minimize electromagnetic interference (EMI) and maintain good signal integrity. These guidelines are especially important for USB high speed and, we recommend they be followed for full and low speed applications as well.

1. D+ and D- must be routed as a differential pair. Match lengths to within 100 mils. D+ and D- must each have a characteristic impedance of 45Ω to GND.
2. It is recommended that the pcb be a minimum of 4-layers. With two routing signal layers (outer layers) and two plane layers (inner layers) with a stackup shown below.



1. Route D+ & D- with a minimum of vias and corners. Always use 45o bends.
2. Route D+ and D- on the top signal layer as much as possible.
3. Do not route D+ & D- under or near crystals, oscillators, clock distribution IC’s, clock signals, switching regulators, magnetic devices, right on the board edge, near mounting holes.
4. Route D+ & D- over a continuous plane layer and never cross splits in a plane.
5. Point-to-point routing should be used for routing D+ & D- between the MCU and the USB connector. If a stub is necessary, keep it less than 200 mils for high speed, and less than 2 inches for full speed.
6. USB high speed D+ & D- must not be routed to more than one USB connector. It is recommended that Full speed USB not be routed to more than one USB connector.
7. Minimum capacitance on USB Vbus = 1uF. This applies to any USB Device, even self-powered ones. This is based on USB Org ECN: Minimum Capacitance Required on VBus, Dec 2008.
8. Maximum capacitance on USB Vbus = 10uF. This is to meet the inrush current specification.