Is Now Part of

ON Semiconductor®

To learn more about ON Semiconductor, please visit our website at
www.onsemi.com

AN-9073
High-Speed USB Switch Layout Considerations

Introduction

This document is intended to aid a systems designer adding a High-Speed (HS) USB device to an application. In most cases, boards are routed using an autorouter that may or may not take high-speed phenomenon into consideration. The end result generally produces non-optimal performance or outright failure. Proper design rules for autorouting can get the best possible result, but even the most sophisticated algorithms still may not produce the desired layout. Hand routing is still commonly used to route sensitive nets in difficult layout conditions. In either case, the layout rules discussed in this application note should be followed.

There are many tradeoffs when designing a circuit board. Usually the type and cost of the application dictate parameters such as size, board material, stackup, and component density... all which play a role in the performance of a high-speed system.

Knowing which nodes are sensitive and may fall victim to high-speed phenomenon is important and, as a result, should be prioritized in a design. The most sensitive nets should be considered first and given priority when placing components and selecting routing layers. Typically, these lines are the high-speed data and clock lines and usually run in the hundreds of megahertz and above.

Transmission Lines

The key of high-speed signals is the transmission line. If a particular net's impedance is known, a transmission line may be constructed to properly transfer the energy from source to destination with minimal loss and to reduce negative side effects, such as crosstalk and EMI. There are many types of transmission line structures to consider, all with tradeoffs that may decide which is best. In many cases, the board stackup dictates the structure. The goal of the transmission line is to transfer energy in a signal with minimal loss using controlled impedance.

Controlling Impedance

Ideally, a signal launched from a source is absorbed at the destination with no changes in wave shape or amplitude. This, of course, is not possible due to physical nature of the connectors, board traces, IC packages, etc. where the impedance of these items is not perfectly matched to the driver or receiver. The reality, is that every component in the path of the signal has characteristics that are not perfectly matched. The result of these imperfections commonly surface as reflections and can negatively impact jitter, crosstalk, and EMI. Reflections may be observed as softening of an edge transition and may produce a reduced eye opening at the device's receiver. These negative effects require that components are strategically placed and transmission lines are routed with care.

Selecting a Transmission Line Structure

The first consideration in selecting a transmission line structure is determining whether the transmission is single-ended or differential. This is important because the common-mode noise rejection characteristics of differential transmissions make it the obvious choice for porting high-speed data. Differential data is best transported in a differential transmission line structure (rather than two single-ended transmission lines). This is typically accomplished using a "coupled" transmission line where the odd-mode impedance is the dominant factor.

Microstrip or Stripline?

In most applications, the choice of transmission line comes down to microstrip or stripline, where microstrip generally exists on the external layers of the stackup (top or bottom) and stripline is buried between layers. In general:

1. Microstrip is less capacitive and therefore less lossy than stripline. It's a good choice for high-speed nets where placing a via in the signal path is unacceptable.
2. Stripline is a good choice for lower EMI and interference from surface components and routes. It is also a better choice for densely populated boards where board real-estate is at a premium. If more routing is required, add more layers.

Guidelines for Successful USB Switch Board Routing

1. USB switches should be placed as close to the USB controller as possible. There should be no more than 1 inch between the controller and USB switch. For best results, this distance should be kept to less than ¼ of the transmission electrical length (<18mm). The distance from USB switch to USB connector should be <1 inch to minimize system signal loss.
2. The D+D- lines should be routed using an impedance calculator to ensure they achieve 90Ω differential impedance. Shorter traces usually mean less loss, less chance of picking up stray noise, and may radiate less EMI.
3. Select the best transmission line for the application. In the case of a densely populated board (such as a laptop computer), an edge-coupled differential stripline would be the right choice.
4. Terminations are critically important in a transmission line and differ among driver and receiver topologies. Some applications require a source or "back" termination, some require receiver or "far end" terminations, and some require both. In all cases, the termination components must be placed as close as possible to their respective pins. For this reason, many high-speed circuits have on-chip terminations to aid in the process.
5. Minimize the use of vias and keep high-speed lines on certain planes in the stackup. Vias are an interruption in the impedance of the transmission line and should be avoided. Choosing a stripline scheme generally forces the use of at least two vias, one on each end to get the signal to and from the surface. Moving to another plane for the convenience of the route should be avoided.
6. If lines must be crossed in a stripline stackup, cross orthogonally to avoid noise coupling (traces running in parallel couple).
7. When space allows, GND copper should be run adjacent to the transmission lines to help isolate from noise sources. The distance from the transmission line to the GND should be considered in the impedance calculation. Anything (GND, power, components, etc.) close to the transmission line could alter the impedance of the transmission line and create a discontinuity.
8. When routing high-speed lines, be aware of trace length. This is especially important for differential traces and parallel data where timing is critical. It is not uncommon to require lengths to be matched to within a millimeter.
9. Avoid sharp bends in PCB traces; a chamfer or rounding is generally preferred.
10. When decoupling power pins, the choice of a low-ESR capacitor placed at or very near the power pin is critical. Where sensitive analog circuits are being powered, a tuned PI filter should be used to negate the effects of switching power supplies and other noise sources.

Related Datasheets

FSUSB30 — Low Power, Two-Port, High-Speed USB 2.0 (480Mbps) Switch

FSUSB31 — Low-Power, Single-Port High-Speed USB 2.0 (480Mbps) Switch

FSUSB40 — Low-Power, Two-Port, Hi-Speed, USB2.0 (480Mbps) Switch

FSUSB42 — Low-Power, Two-Port, Hi-Speed, USB2.0 (480Mbps) UART Switch

FSA221 — USB2.0 High-Speed (480Mbps) and Audio Switches with Negative Signal Capability

FSA321 — USB2.0 Hi-Speed (480Mbps) and Audio Switches with Negative Signal Capability and Built-in Termination on Unselected Audio Paths

FSA9280A / FSA9288A USB Port Multimedia Switch, Featuring Automatic Select and Accessory Detection

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Literature Distribution Center for ON Semiconductor
19521 E. 32nd Pkwy, Aurora, Colorado 80011 USA
Phone: 303−675−2175 or 800−344−3860 Toll Free USA/Canada
Fax: 303−675−2176 or 800−344−3867 Toll Free USA/Canada
Email: orderlit@onsemi.com

N. American Technical Support: 800−282−9855 Toll Free USA/Canada
Europe, Middle East and Africa Technical Support:
Phone: 421 33 790 2910
Japan Customer Focus Center
Phone: 81−3−5817−1050

ON Semiconductor Website: **www.onsemi.com**

Order Literature: http://www.onsemi.com/orderlit

For additional information, please contact your local Sales Representative