OPTi Inc.
888 Tasman Drive
Milpitas, CA 95035
(408) 486-8000
Fax: (408) 486-8001

Application Note

Product Name: FireLink (82C681) PCI-to-USB Bridge

Title: USB Board Level Design Considerations

Date: March 5, 1997

Scope

This application note addresses the PCB layout recommendations for the USB portion of the Firelink (82C861) demo board. For the 82C916 Serial Codec/Mixer design information, refer to Application Note 915-2000-065 Revision 1.1, “82C916 Board Level Design Considerations”.

Discussion

The followings details the PCB layout recommendations of the Firelink (82C861) demo board.

1. Place the 82C861 physically near to the USB connectors as much as possible.

2. When routing the USB signal outputs (both VD1+/ VD1- and VD2+/VD2-) of the 82C861, avoid running the traces near high speed clock lines or other such signals.

3. Noise can be minimized by running pairs of USB signals parallel to each other and running grounded guard traces on each side of the signal pair. It should not have any other signals (except for power and ground) located between the USB signal traces.

4. The USBVCC and ground are subject to noise both from on-board sources and also from power supply. They should be decoupled with ferrite beads. Separate ferrite beads (they should be rated at 500 mA) may be used on each USBVCC line for each USB connector and they should be placed close to the connectors. Ferrite beads should also be used on the ground lines to USB connectors.

5. Maintain a continuous ground plane in the vicinity of the USB connectors.

6. The series termination resistors on the USB signal traces should be placed close to the 82C861. This practice minimizes discontinuity in the USB line impedance. The decoupling capacitors on the USB signal traces provide for bypassing high frequency noise need to be placed between the 82C861 and the series termination resistors so that they do not adversely affect the line impedance.

Figure 1 Sample Add-In Board Placement Diagram