AN-6064 — FSHDMI311 PCB Layout Guidelines

DVI/HDMI™ Repeater

Introduction

This application note provides guidelines for successful PCB layout techniques for Fairchild's FSHDMI311 DVI/HDMI™ Repeater. *For additional information, review the FSHDMI311 datasheet.*

The FSHDMI311 features one HDMI input port and one HDMI output port. It re-transmits the high-speed TMDS signals and the low-speed DDC signals. This guide discusses the PCB placement and routing of all critical components in order of importance. In addition to the FSHDMI311-specific recommendations listed here, common high-speed layout techniques should also be used.

Applications Board Layout

Figure 1 shows the layout of the traces on the applications board (top signal - layer 1). The high-speed TMDS traces on the receive side are marked. The transmit-side TMDS traces are on the opposite side of the board (layer 4).

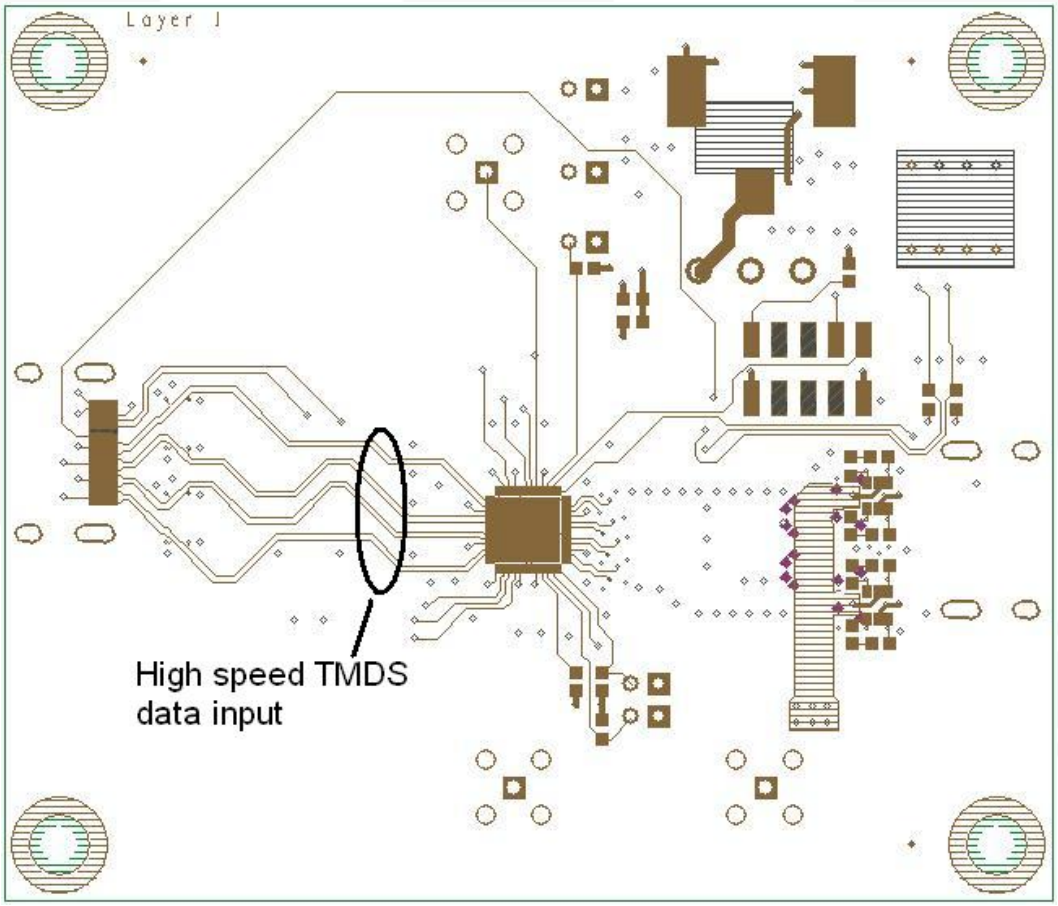

Figure 1. FSHDMI311 Applications Board Receive TMDS Traces

HDMI Connectors

One of the most important aspects of a good PCB design is component placement. HDMI connectors should be placed as close to the FSHDMI311 device as physically possible. The receive-side connector should have priority over the transmit-side connector, as it is most likely to be carrying signals with the worst signal impairments.

TMDS Lines

All TMDS lines should be controlled, 100Ω–differential impedance traces. These traces must be treated as the most critical traces in the PCB layout. Micro-strip traces on the outer layer of the PCB are recommended over strip-line on inner layers as they do not require interlayer vias. If micro-strip traces are used, there should be a continuous reference plane on the layer underneath the traces. If strip-line traces are used, they should be in the PCB stack-up between two continuous reference planes.

All TMDS pairs to a single port should be as short as possible and follow similar paths. Enough space should be left between the differential pairs so that the N+ of one pair does not couple to the P- of another pair. One possible method is to make the inter-pair distance at least four to five times the intra-pair spacing. (An impedance calculator can help determine optimum geometries.)

Depending on the PCB stack-up, the signal propagation velocity along typical FR4, or similar substrate, is ~50ps/cm. Rise-times in HDMI are on the order of 100ps, so all traces of a single port should be matched in length to less than 2mm. Where possible, use trace widths equal to or as close as possible to the device solder pad widths. This can be achieved by adjusting the PCB stack up appropriately and avoids the pads themselves becoming transmission line discontinuities. 90-degree turns should be avoided; use smooth curved lines (if CAD tool allows) or combinations of 45-degree turns.

Power Supplies

The FSHDMI311 device has two power domains: V_{DD} for digital circuits and low-speed I/Os; and AV_{DD} for analog circuits and high-speed I/Os. Using separate power planes for each domain is recommended. (If separate power planes cannot be used, the use of PI filters is recommended.) A single ground (GND) plane should be used and the PCB stack-up between power and ground kept as small as possible to maximize the interlayer decoupling capacitor formed between them. Figure 2 shows the different power planes on layer 3.

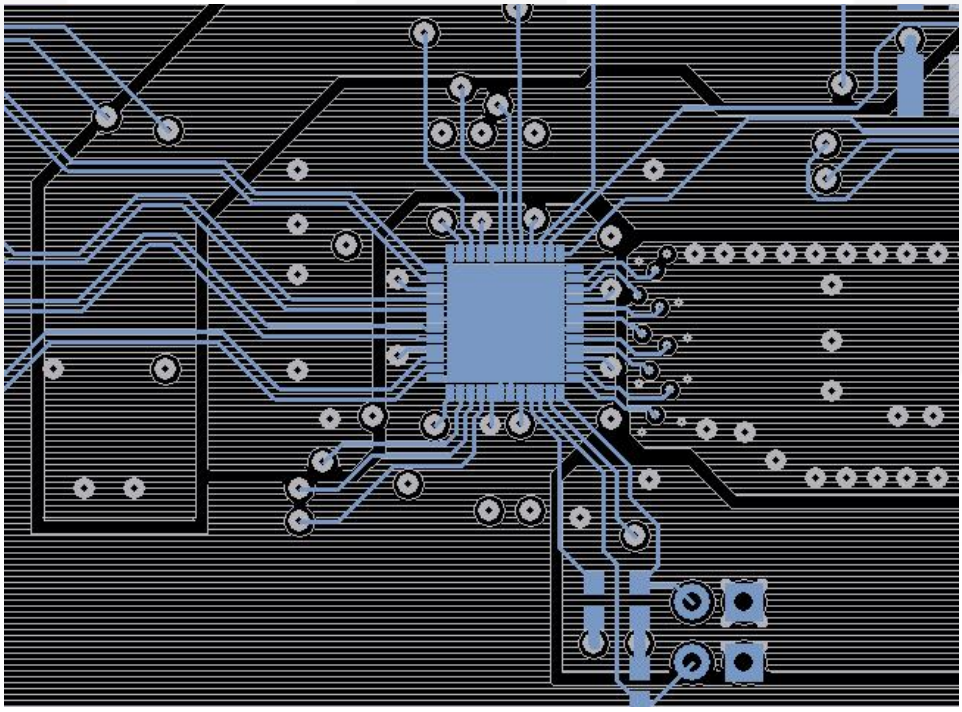

Figure 2. FSHDMI311 Applications Board Power Planes

Decoupling Capacitors

All V_{DD} and AV_{DD} pins should have a 0.01µF decoupling capacitor connected between the pin and a ground plane. These capacitors should be placed as close possible to the respective power and ground pins of the device. In general, this means placing them on the bottom side of the PCB, underneath the actual device. When placing on bottom side of PCB, ensure adequate (i.e. large or multiple) interlayer vias are used for both the pin side and ground-plane side. Make the leads to these capacitors as short as possible to reduce parasitic inductance effects.

Figure 3 displays the bypass capacitors underneath the FSHDMI311 device.

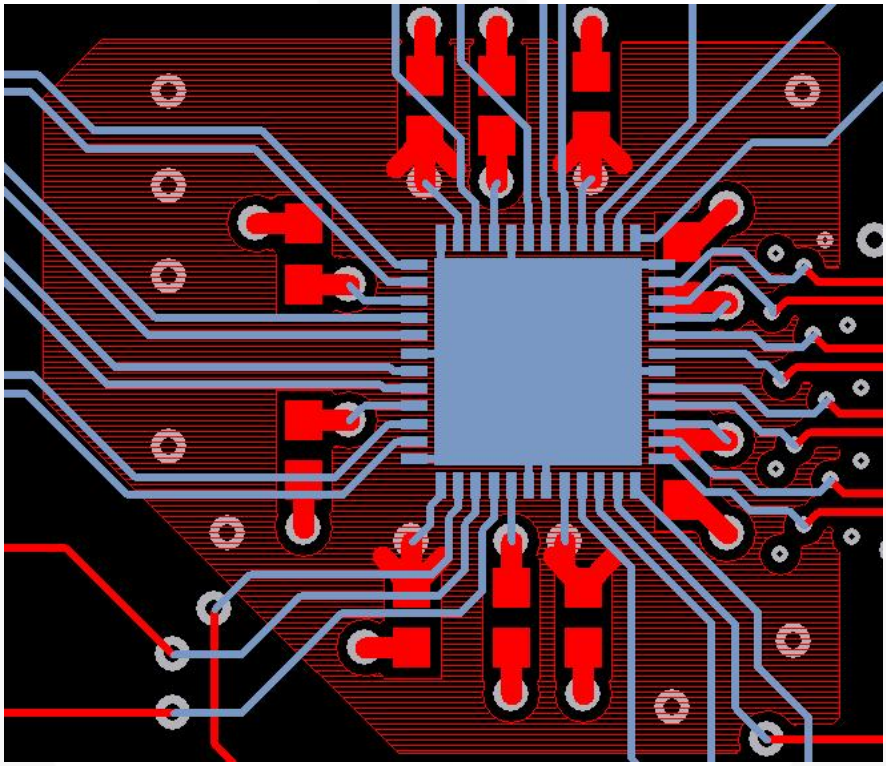

Figure 3. FSHDMI311 Applications Board Power Bypass Capacitors

External ESD Protection

To achieve the 4kV contact / 8kV air required for consumer devices, it is normally necessary to use additional ESD protection devices in the application circuit. These should be placed to have minimum impact on the high-speed TMDS signals. The optimum placement for these protection devices is bridging the TMDS traces as close to the device (I/O connector) input pins as possible. If this is not possible (due to fine pitch and number of TMDS traces), they can be placed on the bottom side using the same precautions as used for decoupling capacitors. In receiver applications, the receiver side requires such protection; in a cable extender application, both receive and transmit sides of the device should be protected.

VSADJ Resistor

The VSADJ resistor should be placed physically close to the device, on component side, where possible. Avoid any high-speed lines nearby and ensure good ground connection using multiple interlayer vias.

DDC Lines

Layout of DDC lines is not as critical as TMDS lines, but still should be treated with some care. The main constraint is to avoid any unnecessary extra capacitance, particularly where pull-up resistors are not being used (the FSHDMI311 device has an internal pull-up that may be sufficient in certain applications). The recommendation for these lines is to keep them as short as possible. If long runs cannot be avoided, use fine trace widths and, if necessary (where capacitance is still a problem), remove the V_{DD}/GND plane from underneath.

HDMI, the HDMI logo, and High-Definition Multimedia Interface are trademarks or registered trademarks of HDMI Licensing LLC.