

PI3HDMIxxx 4-Layer PCB Layout Guideline for HDMI Products

Introduction

The differential trace impedance of HDMI is specified at $100\Omega \pm 15\%$ in Test ID 8-8 in HDMI Compliance Test Specification Rev.1.2a and 1.3a. Rev. 1.3a has a more relaxed specification which allows an occurrence of a single excursion out to a max/min of $100\Omega \pm 25\%$ and of a duration less than 250ps. Generating PCB layout with care can compensate or prevent from significant deviation from desired impedance. General PCB knowledge (PCB Prepreg Selection, Critical Signal Trace Length and Microstrip Stub Effect) and impedance control on 4-layer PCB of 1080+2116 Prepreg will be discussed in this layout guideline.

PCB Prepreg Selection

Although 4-layer FR4 PCB with 2116 (high-density) Prepreg is popular for PC motherboard, it will be costly if impedance control is required and its insertion loss is not acceptable. Therefore, it is not recommended in HDMI applications. Instead, Prepreg of 1080+2116 (mid-density) or 2116+7628 (low-density) is recommended. With different Prepregs, trace width and spacing have to be adjusted accordingly. Recommended dimensions are stated below.

For 1080+2116 Prepreg:

(a) Stack Up

Signal	1.9 mil
Prepreg 1080+2116	7.3 mil
Gnd	1.2 mil
Core	44 mil
Power	1.2 mil
Prepreg 1080+2116	7.3 mil
Signal	1.9 mil

(b) Isolation Spacing

30 mil

(c) 100 Ω Differential Trace Width/Spacing

W/S = 8.0 / 9.0 mil
or 9.0 / 11 mil for long traces

(d) 50 Ω SE Trace Width

W = 12 mil

For 2116+7628 Prepreg:

(a) Stack Up

Signal	1.9 mil
Prepreg 2116+7628	11.6 mil
Gnd	1.2 mil
Core	31 mil
Power	1.2 mil
Prepreg 2116+7628	11.6 mil
Signal	1.9 mil

(b) Isolation Spacing

45 mil

(c) 100 Ω Differential Trace Width/Spacing

W/S = 8.5 / 6.5 mil
or W/S = 12 / 7.0 mil for long traces

(d) 50 Ω SE Trace Width

W = 20 mil

Table 1: PCB Prepreg Recommendation

Normally, PCB manufactories control width and spacing of traces at ± 1 -mil tolerance. At the locations of HDMI connector, components and IC devices, the tolerance is best adjusted to ± 0.5 mil so as to reduce deviation. The trade off is cost.

Critical Signal Trace Length

To prevent from signal reflection, signal trace length cannot be longer than the following two critical length limitations:

- (a) 1/16 wavelength of Signal, λ ; the relationship between signal wavelength and signal frequency is defined as

$$f = \frac{c}{\lambda \sqrt{\epsilon_R \mu_R}}$$

where $\epsilon_R = 4.3 \sim 4.7$ and $\mu_R \sim 1$ for FR4 material.

For a 1.25GHz signal running on FR4 material, critical length limitation is determined as
Critical Length < $(1/16)\lambda \cong 280$ mil;

and

- (b) 1/3 length of rising edge of signal; the length, l , is defined as

$$l = \frac{T_r}{D}$$

where l = length of rising edge of signal in inch
 T_r = rise time of signal in ps
 D = delay in ps/inch

The delay of FR4 material is 180 ps/inch. As T_r for HDMI signal is 200ps, signal speed cannot exceed 370 mil which is derived from

$$\text{Critical Length} < \frac{\frac{1}{3} \times 200 \text{ ps}}{180 \text{ ps/in}} \times 1,000 \text{ mil/in} = 370 \text{ mil.}$$

If the signal traces are long, it is recommended to use wider differential trace width and spacing since the impedance is more consistent and easier to be controlled if the width is increased. Please refer to Table 1 for detailed width and spacing selection.

Microstrip Stub Effect

As stub adds reactance to PCB trace and lowers the trace impedance, it is not preferable to create any stub on HDMI layout. If an open stub is $1/2$ wavelength long, it seems to have a shunt capacitor shorted to Ground on the trace. Likewise, if a short stub is $1/2$ wavelength long, it seems to add an inductor or an open on the trace.

Open Stub

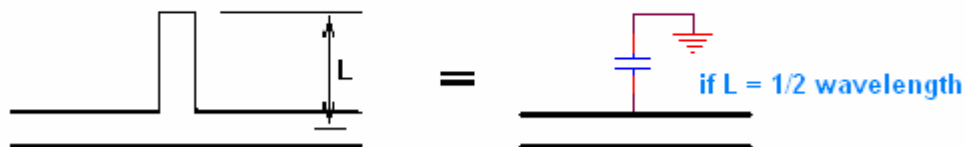


Figure 1: $1/2\lambda$ Open Stub

Short Stub

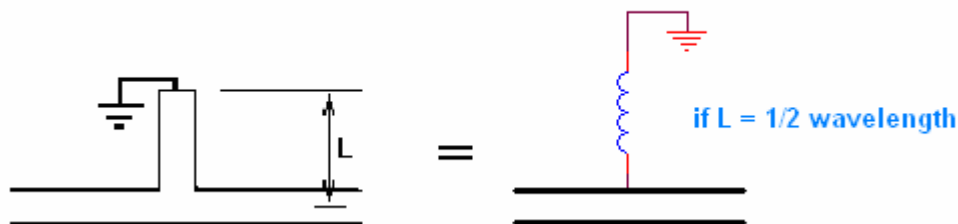


Figure 2: $1/2\lambda$ Short Stub

If it is unavoidable, stub must be kept shorter than $1/6$ length of rising edge of signal. In practice, for a 200-ps HDMI signal, the length of stub cannot be longer than $1/6 \times 200\text{ps} = 33\text{ps}$.

Compensation of Solder Pad / Via

Solder pad and via create discontinuities of traces. As a result, the trace impedance will be lowered. Creating holes on ground plane of PCB under these components can minimize the capacitance between pad/via and ground plane so as to compensate the impact on trace impedance. Dimensions of holes are recommended in (i) – (iv) in Section (A).

The HDMI connector pads are designed to interact with connector pins so as to produce desired trace impedance and create robust signal paths. In order to achieve these, the reference plane, which is the ground plane for HDMI connector in the example in (v) in Section (A), should be removed immediately under the solder pads.

A case study of restricted PCB area is provided in Section (B) for reference.

Section (A): Recommended Dimensions of Holes on Ground Plane

A PCB of 1080+2116 Prepreg is used as an example below. Differential trace width and spacing of 8.0 and 9.0 mil are employed. The dimensions of holes on the ground layer are drawn.

- (i) Under 0603 ESD / Pull-up Resistor Pads

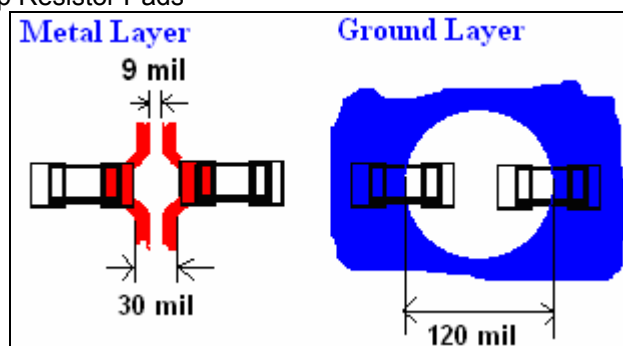


Figure 3: Layout Recommendation of ESD / Pull-up Resistors in 0603 Package

- (ii) Under 0402 ESD / Pull-up Resistor Pads

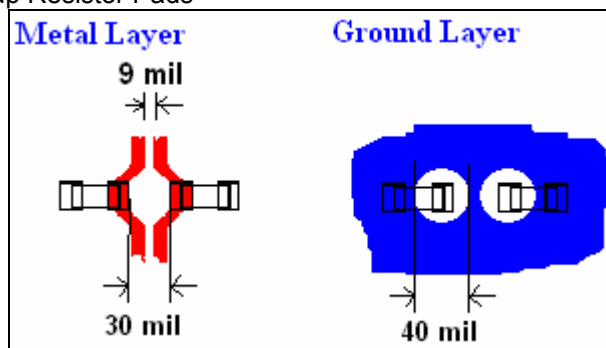


Figure 4: Layout Recommendation of ESD / Pull-up Resistors in 0402 Package

(iii) Under Solder Pads of HDMI Switch, e.g. TQFN-56 below

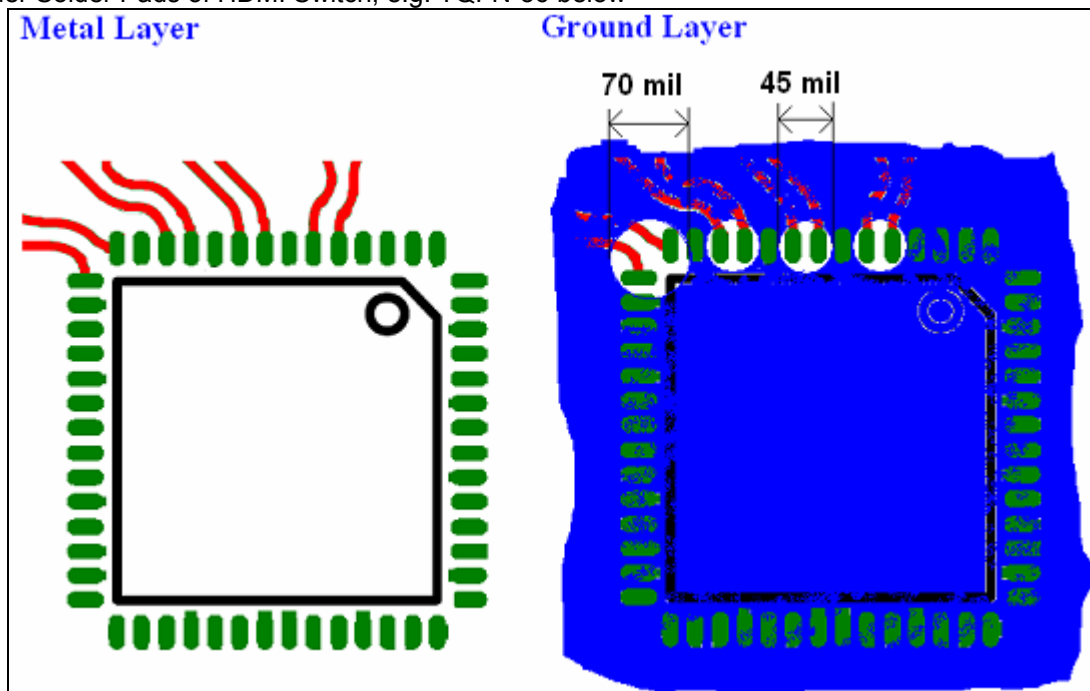


Figure 5: Layout Recommendation of HDMI Switch

(iv) Under Via

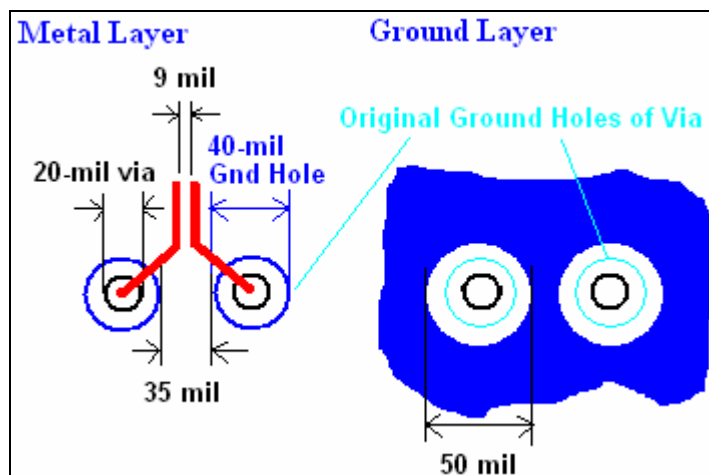


Figure 6: Layout Recommendation of Via

(v) Under HDMI Connector Pads of 3 High-speed Signal Pairs and 1 Clock Pair

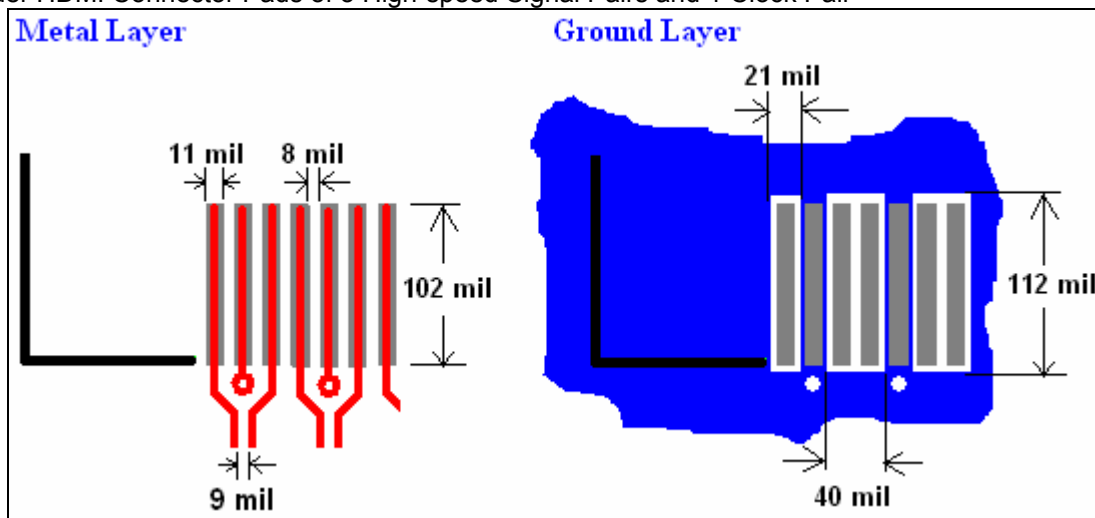


Figure 7: Layout Recommendation of HDMI Connector

Section (B): Case Study of Restricted Area on PCB

In practice, spacing on PCB is often a concern on layout; also, vias and stubs are unavoidable to connect ESD components and/or pull-up resistors on the bottom metal layer of PCB. A PCB including ESD components, vias and pull-up resistors is provided for your reference below.

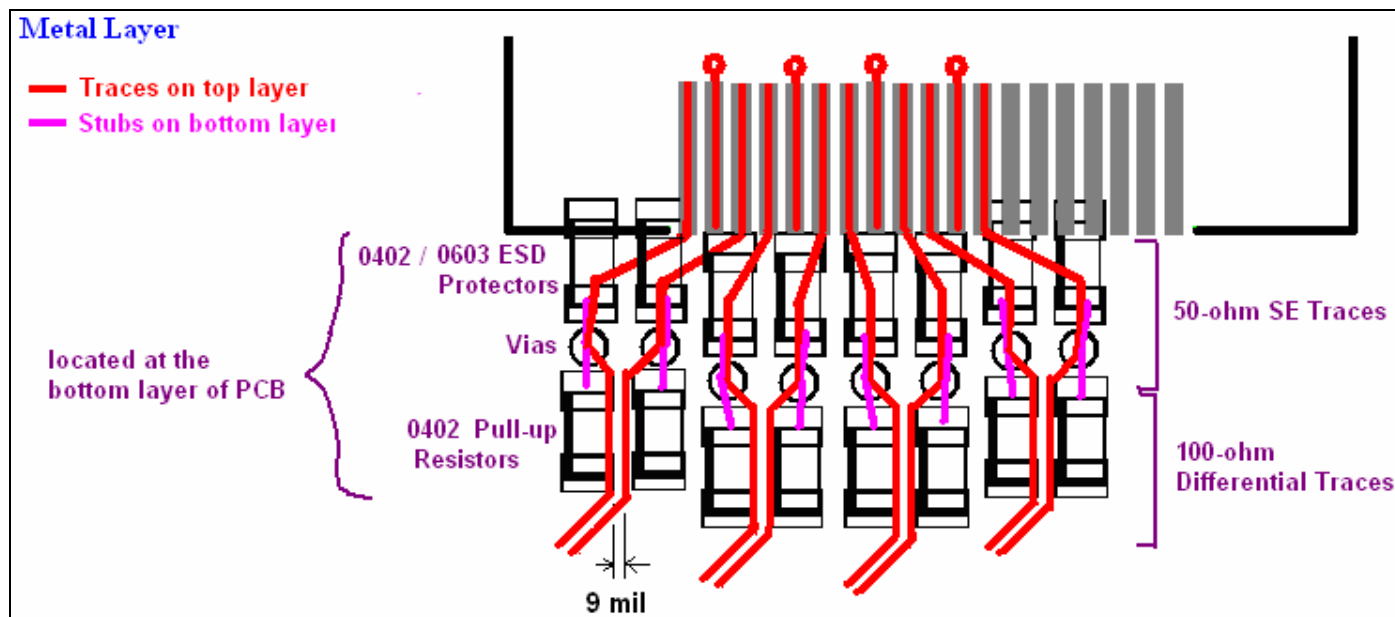


Figure 8: Top Metal Layout Recommendation of PCB with ESD + Via + Pull-up Resistors

Recommendations:

- (a) Place vias as close to HDMI connector as possible
 - When signals travel from HDMI connector to HDMI solder pads, the impedance is increased because of dielectric change. This increase can help compensating the impedance drop introduced by vias if the vias are located close to HDMI connector.
 - As the vias are close to HDMI connector, there may not be enough space to route 100Ω differential traces between HDMI connector and vias. 50Ω single-ended traces can be employed there, instead, provided that the distance is short.
- (b) Remove ground plane under solder pads of HDMI signal and clock pairs
- (c) Use passive components (pull-up resistors and/or ESD protectors) with small package
 - 0402 package instead of 0603 is recommended as smaller solder pads have less impact on trace impedance.
- (d) Use 9.0/11 mil as trace width and spacing of differential signal traces
 - Impedance is more controllable if trace is wider.
 - Appropriate PCB prepreg must also be followed such that the trace impedance is within impedance tolerance. Please refer to PCB Prepreg and Impedance Tolerance Sections for details.
- (e) Use stubs as short as possible
 - The stubs between ESD protectors and vias, and vias and pull-up resistors must be shorter than 1/6 length of rising edge of signal as described in Microstrip Stub Section.
- (f) Remove ground plane under resistor/ESD pads and vias
 - The hole must be big enough to cover all ESD protector pads, vias and pull-up resistor pads for all four pairs of signals at one HDMI connector. Please refer to Figure 9.

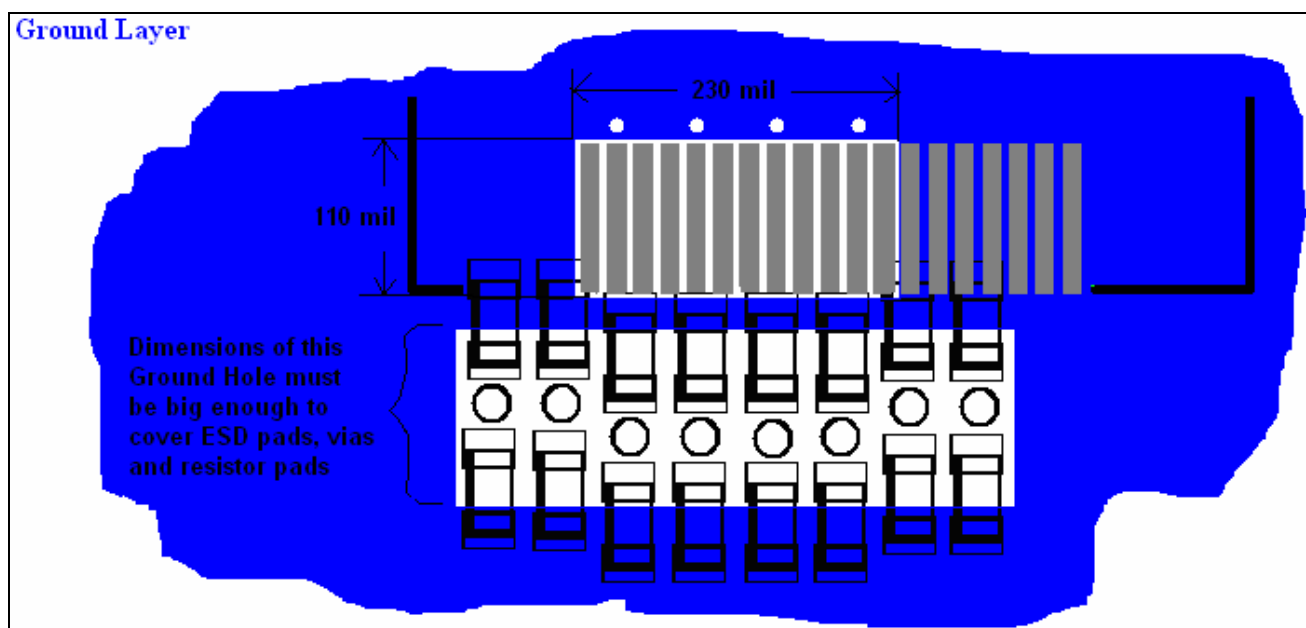


Figure 9: Ground Layer Recommendation of PCB with ESD + Via + Pull-up Resistors