大联大控股
A Member of WPG Holdings
世 平 集 团
World Peace Industrial Group

(連結)世平安森美半導體 Redriver Layout 注意事項

Jun Kuo
2020.12.30

Agenda

- **Guidelines & Examples: Transmission Line Layout Structure**
  - Single-Ended
    - Trace Angle
    - Trace Length
    - Routing Around Components
  - Differential Pairs
    - Trace Angle
    - Trace Length & Width
    - Routing Around Components
    - Trace Spacing
  - Routing for Surface Mounted Devices
  - Stackup
  - Vias & Via Stubs

TRANSMISSION LINE STRUCTURE GUIDELINES

Single-Ended

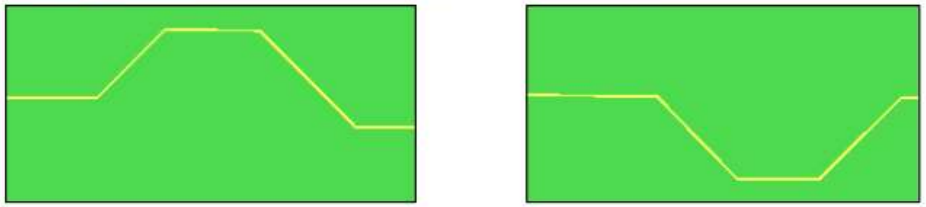
Single-Ended: Trace Angles

Guidelines for Trace Angles

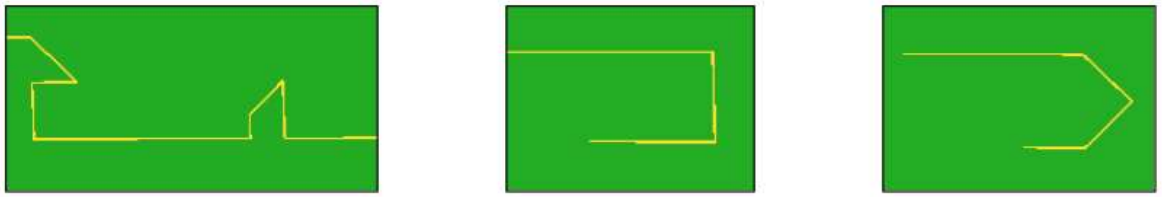
- Any tight bends or 90° angles in the transmission line layout should be avoided since electromagnetic interference (EMI) and reflections can occur. And as a result can significantly contribute to signal integrity degradation for high frequency lines.

Examples:

Acceptable Transmission Line Routing:



Not-Acceptable Transmission Line Routing:

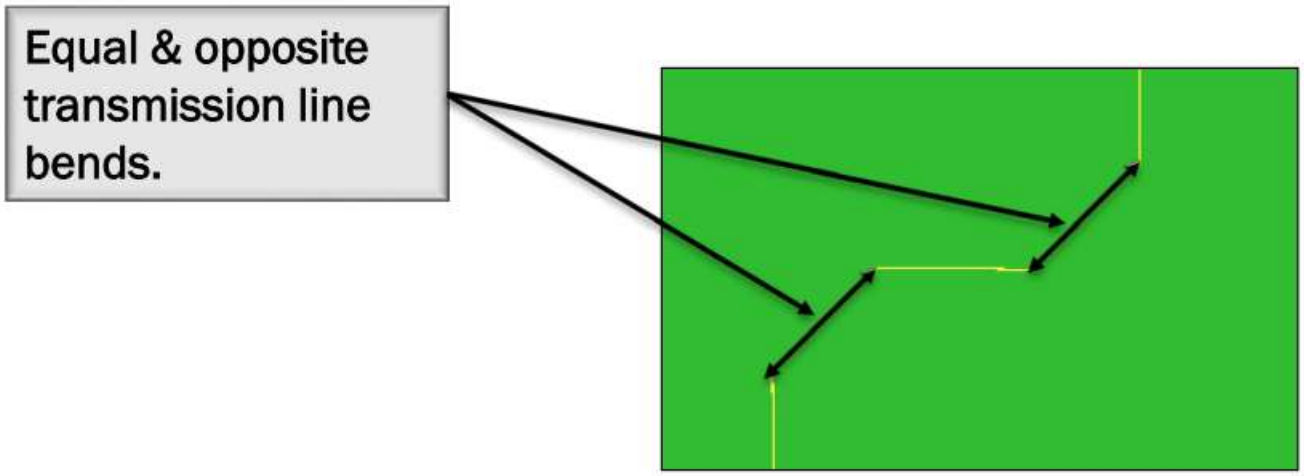


Single-Ended: Equal & Opposite Trace Angles

Equal & Opposite Trace Angles

- Equal and opposite transmission line bends can counteract each other which can be beneficial to the overall signal integrity.

Example:



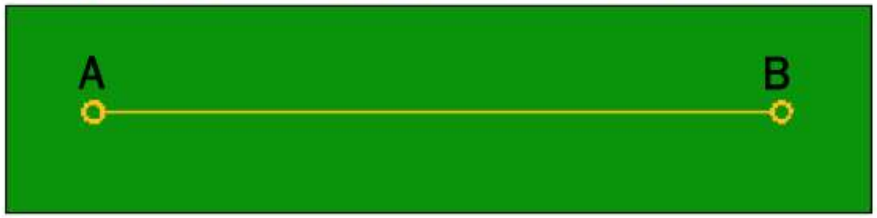
Singled-Ended: Trace Length

General Guidelines for Trace Length

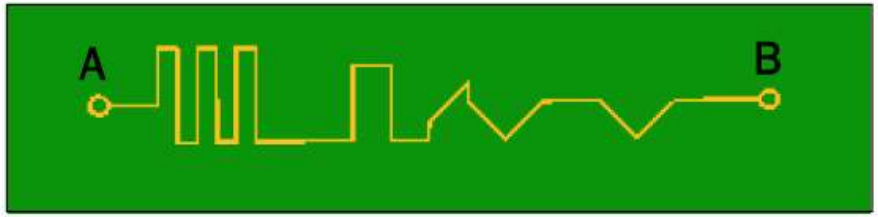
- The greater the trace length the more parasitic losses there are associated with the trace. Therefore total length should be kept to a minimum.

Examples:

Acceptable Routing for Minimizing Trace Length:



Not-Acceptable Routing for Minimizing Trace Length:



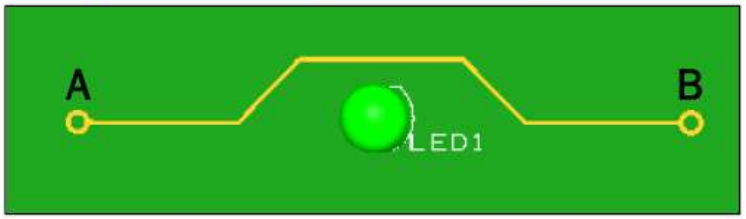
Singled-Ended: Routing Around Components

Guidelines for Routing Around Components

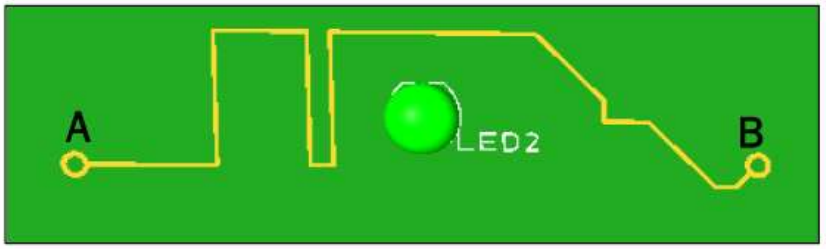
– If the transmission lines must be routed around components, tight bends and trace length guidelines should be implemented to minimize the integrity degradation of the high-frequency signals.

Examples:

Acceptable Routing Around Components:



Not-Acceptable Routing Around Components:



TRANSMISSION LINE STRUCTURE GUIDELINES

Differential Pairs

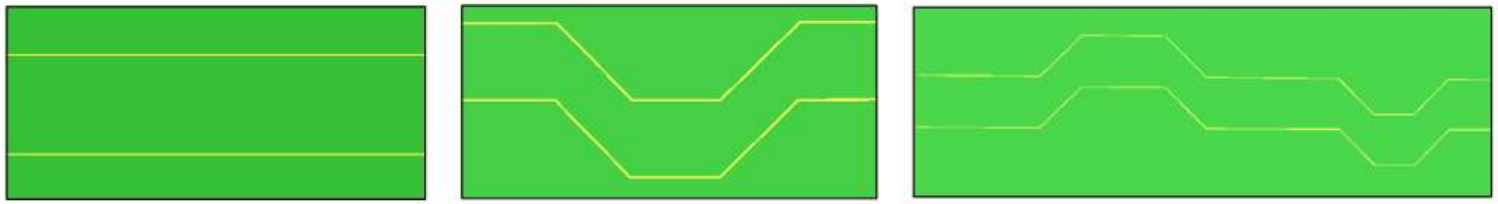
Differential Pairs: Routing

Differential Pairs Routing Guidelines

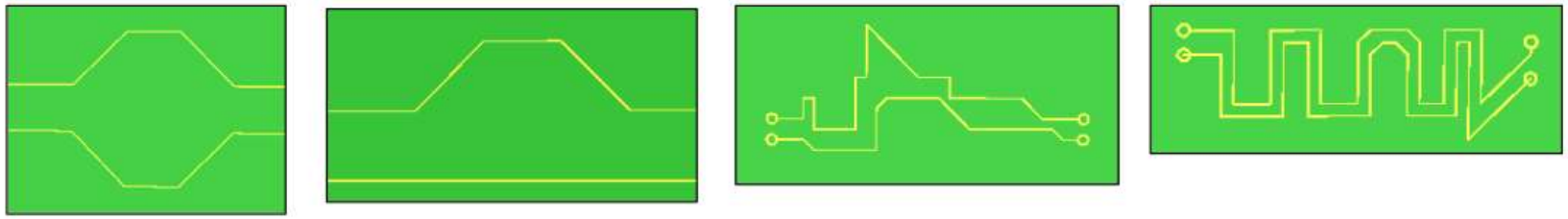
– Tight bends and 90° angles along with different length traces for differential pairs should be avoided since they can significantly contribute to signal integrity degradation for high frequency lines.

Examples:

Acceptable Routing for Differential Pairs:



Not-Acceptable Routing for Differential Pairs:



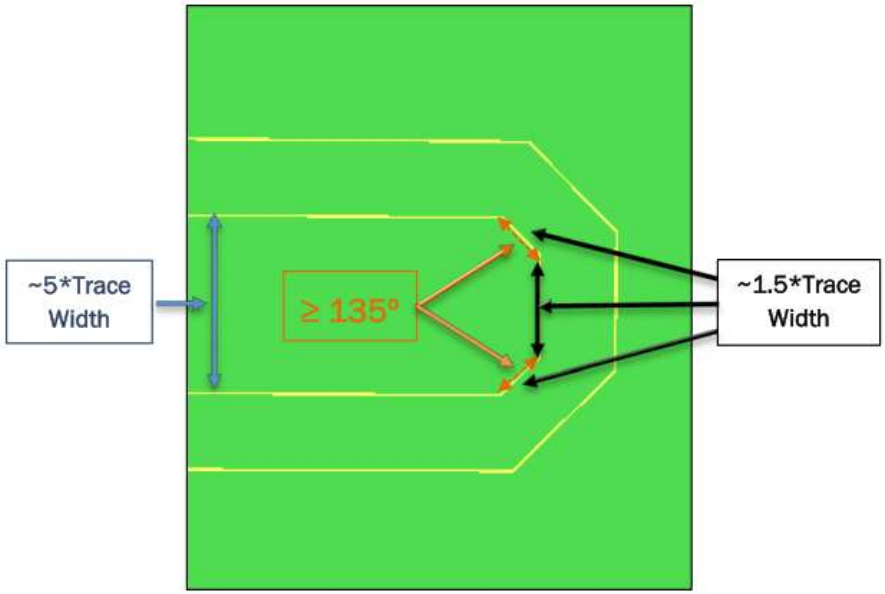
Differential Pairs: Trace Angle

Routing Differential Pairs Guidelines

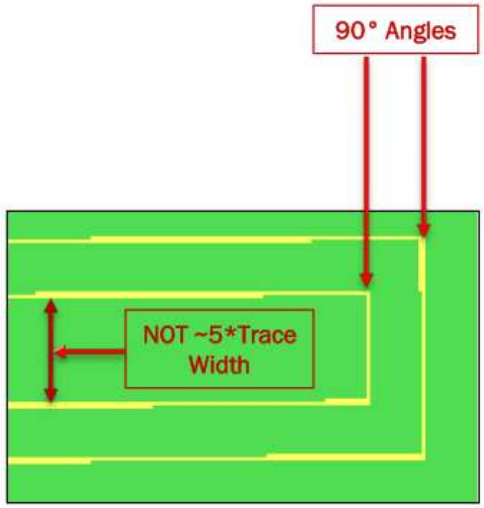
– If corners are required tight bends and 90° angles for differential signal pairs should be avoided.

Examples:

Acceptable Differential Corner Routing:



Not-Acceptable Differential Corner Routing:



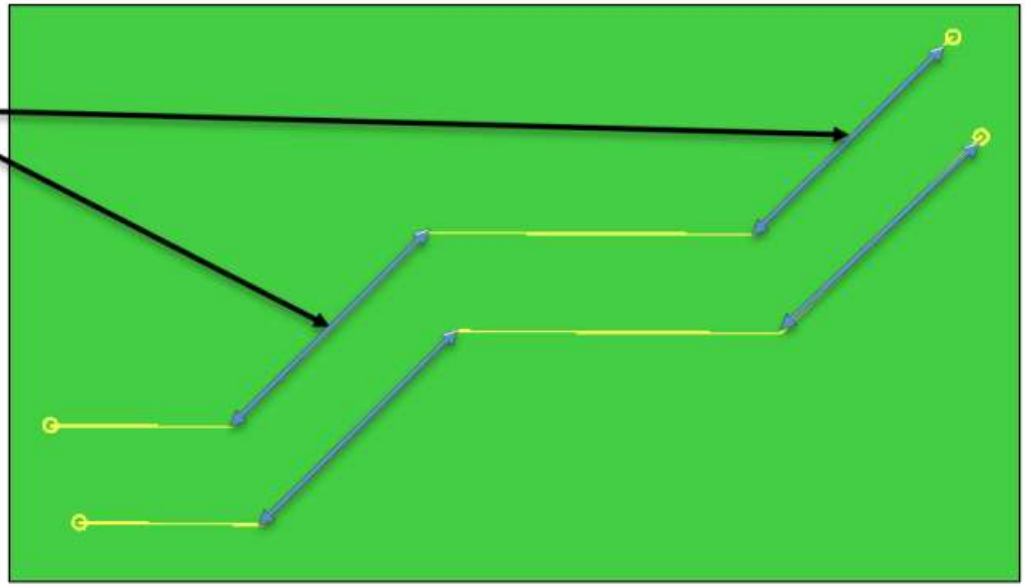
Differential Pairs: Equal & Opposite Trace Angles

Equal & Opposite Trace Angles

– Differential pairs can also benefit from having equal and opposite transmission line bends to counteract each other.

Example:

Equal & opposite transmission line bends on differential pair.



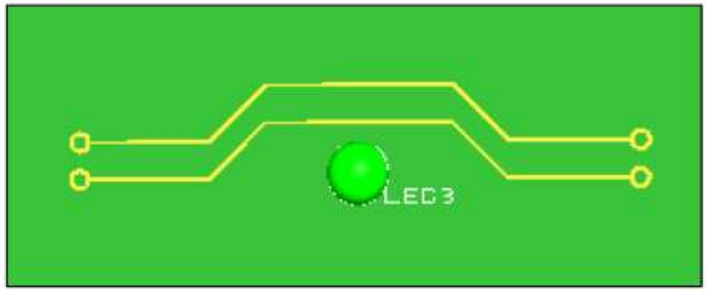
Differential Pairs: Routing Around Components

Trace Length Guidelines for Differential Pairs

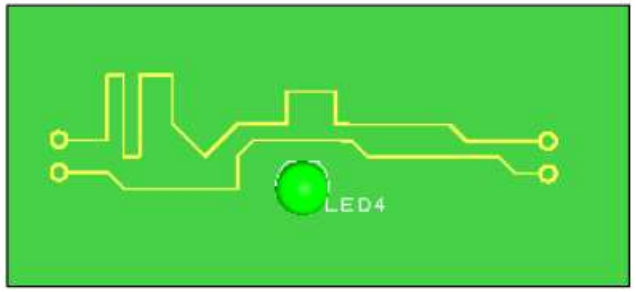
- If differential pairs must be routed around components, tight bends and trace length guidelines should be implemented to minimize the integrity degradation of the high-frequency signals.

Examples:

Acceptable Routing to Match Diff. Pair Lengths:



Not-Acceptable Routing to Match Diff. Pair Lengths:



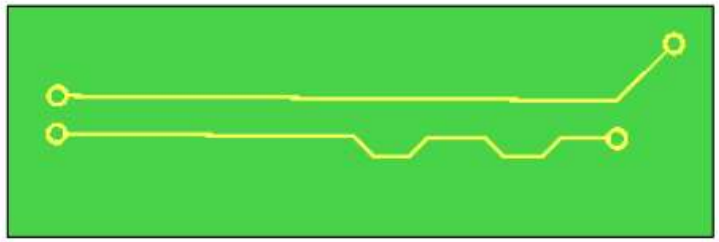
Differential Pairs: Matching Trace Lengths

Matching Differential Pairs Lengths Guidelines

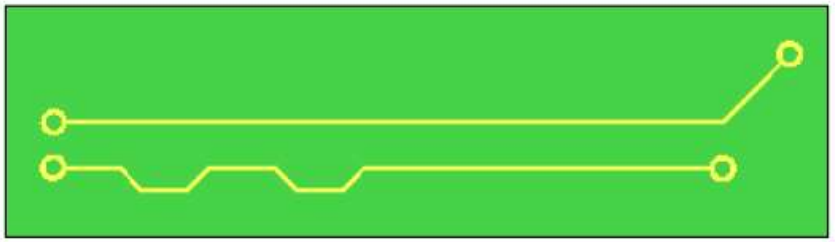
- The length difference between differential pairs should **not** exceed ~5 mils although ideally they should be perfectly matched.
- Sometimes the layout requires the individual lines of the differential pairs to be routed differentially resulting in one transmission line having a greater length then the other. Therefore, it is sometimes necessary to add length to one of the transmission lines by adding additional bends as close as possible to the mismatched ends.

Examples:

Acceptable Routing to Match Diff. Pair Lengths:



Not-Acceptable Routing to Match Diff. Pair Lengths:

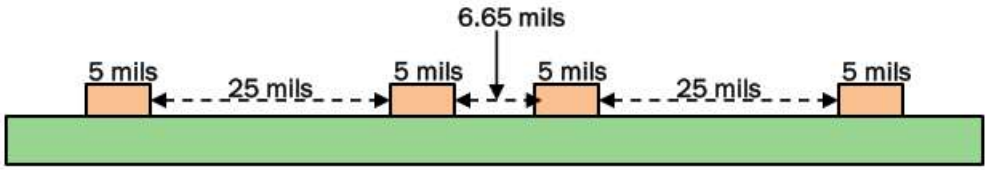


Differential Pairs: Trace Spacing & Width

Differential Pair Spacing & Width Guidelines

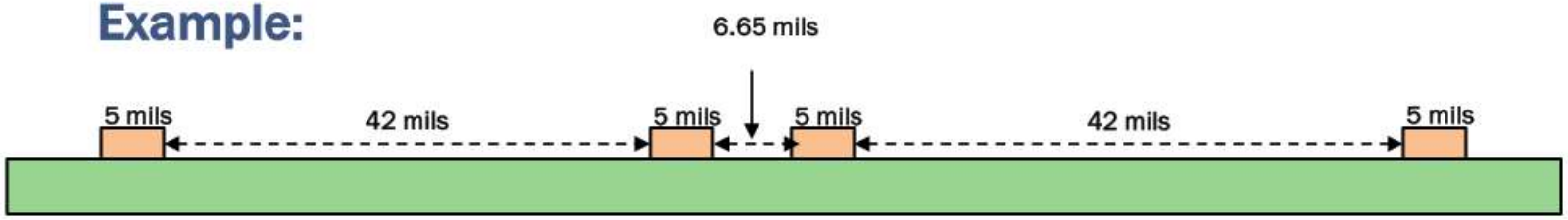
– Differential pairs with other signals present next to them should have at least ~5*trace width spacing. In addition the spacing between the positive and negative differential pairs be ~1.33*trace width.

Example:



– Additional spacing is required between differential pairs and other signals that are periodic or are clock signals. In general the spacing should be ~8.5*trace width.

Example:



TRANSMISSION LINE LAYOUT GUIDELINES

Routing for Surface Mounted Devices

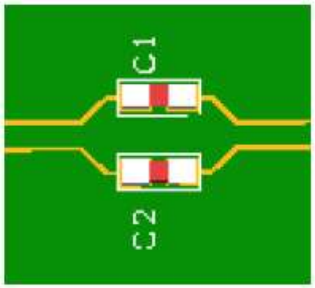
Surface Mounted Devices

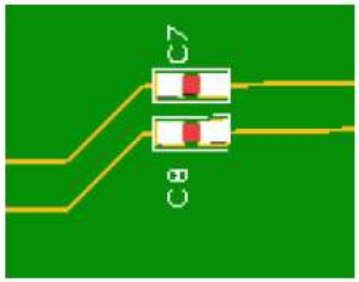
Routing for Surface Mounted Devices Guidelines

- Surface mounted devices (SMDs) should be avoided if possible since they can cause reflections and overall signal degradation specifically for high-speed transmission lines. However, some high-speed applications require SMDs to be used and therefore specific layout guidelines should be used to reduce signal degradation as much as possible.

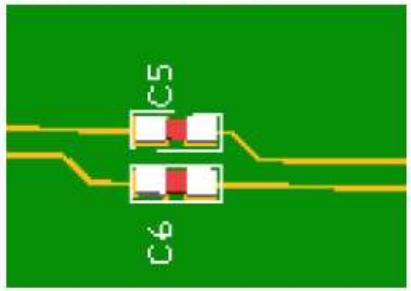
Examples:

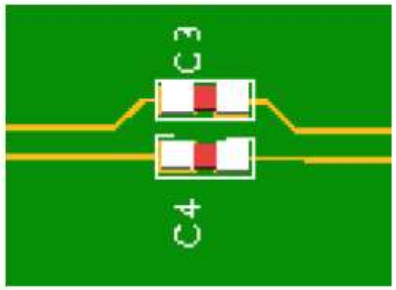
Acceptable Routing of SMDs:





Not-Acceptable Routing of SMDs:





TRANSMISSION LINE LAYOUT GUIDELINES

Stackup

PCB Stackup Characteristics & Guidelines

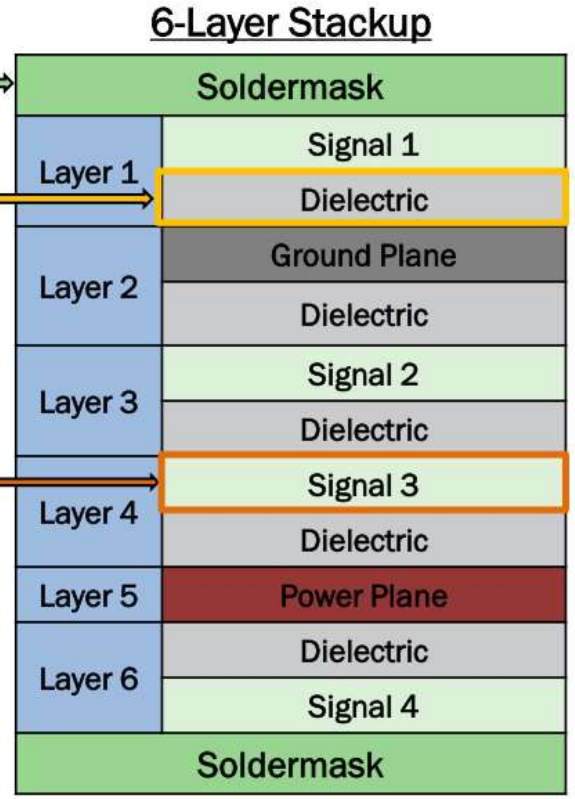
Regardless of the number of layers that are required in a specific stackup, the same principles that are shown in the example of the 6-layer stackup below can be applied.

Example of 6-Layer Stackup:

The soldermask will reduce impedance by ~1-3Ω **only** if the traces are relatively thin.

- FR4 material is commonly used for the dielectric material in board stackups.
- Large dielectrics between the signal layers create less plane capacitance between the layers, however they should be avoided since it is not beneficial for overall signal integrity.

- Signal layers should be placed as close to their adjacent planes. This will improve the electromagnetic interference (EMI), reduce the crosstalk between the traces, and create desirable characteristic impedance (which should be ~50Ω.



TRANSMISSION LINE LAYOUT GUIDELINES

Vias & Via Stubs

Vias: Blind Vias & Vias Stubs

General Vi Chacterisitics & Guidelines:

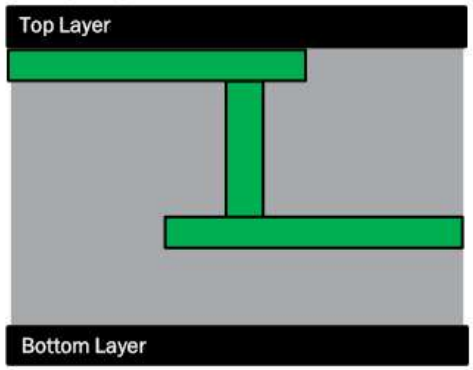
- If possible the use of vias should be avoided since they introduce discontinuities. As a result vias can cause capacitance and inductance which can contribute to reflections and signal degradation.

Minimizing Via Discontinuities

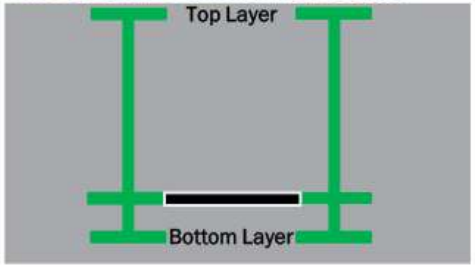
- Blind vias which don't result in any stubs, should be used when possible.
- If vias that go through the entirety of the board are required, the via stubs should be kept to a minimum to mitigate the discontinuities. Via stubs should be less than ~15 mils.

Examples:

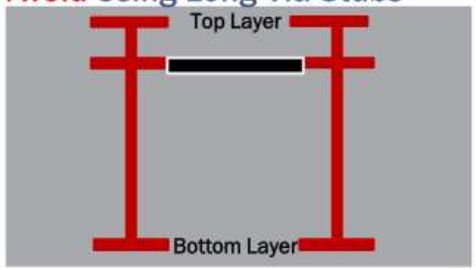
Blind Via



Acceptable Short Via Stubs



Avoid Using Long Via Stubs



大联大控股
A Member of WPG Holdings

世平集团
World Peace Industrial Group

Thank you

产业首选 · 通路标杆