Via spacing on high-performance PCBs

Steve Hageman - February 06, 2013

In the last twenty years multilayer PCBs have become nearly a must for all electronic designs. This is for several reasons:

- Higher density pins and pin count on ICs has increased the routing density of our designs.
  Twenty years ago there were no QFNs and 144-pin packages in everyday use.
- The speed at which our circuits operate has increased probably four fold.
  Twenty years ago RF was commonly defined as everything below 1 GHz; now RF is everything below 6 GHz.

These speeds and densities have driven PCB design also. Where 20 years ago we may have only used microstrip for our RF traces, today we use microstrip, co-planar waveguide and stripline in our multilayer designs [1][2].

In RF design we typically need to work with only the fundamental frequency of operation. For instance: In a 2.4-GHz RF design the goal is to have a nice 2.4-GHz sine wave on our board with low harmonics. The frequencies that we need to be concerned with are really 2.4 GHz.

In digital design, the goal is to have a nice square wave on our board. A 1-GHz digital data signal needs to be square for a good eye diagram. This means that the true operating bandwidth of these traces needs to be at least five times the fundamental frequency or 5 GHz. This is from the rule of thumb that for a good-quality digital signal you need to pass at least the 5th harmonic of that signal.

Naturally there are exceptions to these two rules, but for the majority of the signals and designs we encounter they hold true.

At 6 GHz, a wavelength on FR-4 PCB material is only about 1 inch. With the speeds and frequencies we work with today we can easily get a wavelength or more on our board, so via spacing starts to be an important question.

What is the proper via spacing for a PCB and what happens if we get it wrong?

**Ground Vias**

Multilayer PCBs, especially when designed for RF, may have many layers of ground; even the signal layers typically have ground flooded on them. These grounds need to be all tied together to keep them from acting like stubs or transmission line segments themselves. Or put another way, they need to be a low impedance at all the operating frequencies of the PCB, otherwise they don't look like ground anymore - they start looking reactive to the signals on them.

Any piece of copper, if it isn't terminated in its characteristic impedance, can resonate at 1/4 of a wavelength of the frequency of the current flowing in it (and every 1/2 wavelength after that).
Typically we control this by placing stitching vias around the ground planes at some interval. But what is that interval?

I’ve seen little FM transmitter modules operating at 100 MHz that had vias spaced 30 mils apart, and I’ve personally built 1-GHz RF circuits that have via spacings of 250 mils (see Figure 1). Clearly the 30-mil spacing of the FM transmitter will work, since it is way closer than 1/4 of a wavelength – but is it overkill?

**Figure 1:** A section of a 1-GHz PLL synthesizer used in a 1-GHz receiver. The small grey grid dots are spaced at 0.05 inches. The ground vias (yellow circles) are spaced at about 250 mils on average. This circuit operates perfectly well with this via spacing with no signs of ground impedance issues.

**Spacing Calculations**

Years ago I heard the rule of thumb: “If you space your ground vias at 1/8 of a wavelength or less your ground plane will look like a solid ground.”

This is based on the general RF Stub Length principle: “A stub on a trace starts to become a problem as it approaches 1/8 of a wavelength.” A stub is any piece of copper that extends away from a terminated trace [3].

The ECL guys of the 1970s knew this well [4] and it is a good rule of thumb for general purpose work [5].

An effective stub can also form on a ground plane. On a section of copper ground plane that is terminated at both ends with ground vias but not in the middle. The impedance of that middle section can actually look very reactive as the length of the spacing between the grounding vias exceeds 1/8th of a wavelength.
On our example FR-4 PCB operating at 6 GHz, a wavelength may be calculated from the well known formula as,

$$\lambda = \frac{300}{F \times \sqrt{\varepsilon_r}}$$

Where $F = \text{MHz}$, $\lambda$ (wavelength) = meters, $\varepsilon_r = 4.4$ (for typical FR-4 PCB material [6]).

In tabular form for FR-4 PCB material, 1/8 of a wavelength is approximately,

<table>
<thead>
<tr>
<th>Frequency</th>
<th>Length (Inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 MHz</td>
<td>70</td>
</tr>
<tr>
<td>30 MHz</td>
<td>23</td>
</tr>
<tr>
<td>100 MHz</td>
<td>7</td>
</tr>
<tr>
<td>300 MHz</td>
<td>2.3</td>
</tr>
<tr>
<td>1 GHz</td>
<td>0.7</td>
</tr>
<tr>
<td>3 GHz</td>
<td>0.23</td>
</tr>
<tr>
<td>6 GHz</td>
<td>0.12</td>
</tr>
<tr>
<td>10 GHz</td>
<td>0.07</td>
</tr>
</tbody>
</table>

**Figure 2: Calculated 1/8 of a wavelength for various frequencies on FR-4 PCB material.**

Figure 2 shows an interesting result: For example - for a small 2x2-inch FM transmitter PCB operating at 100 MHz, a single via in each corner of the board will be way less than 1/8th of a wavelength! And the board will work fine from an RF point of view!

**There is Always a Caveat Somewhere**

There are a few circuit caveats to take note of however: In analog design, and especially circuits designed with broadband amplifiers and other broadband parts with gain, we may design a 1-GHz circuit that may contain amplifiers that have bandwidths that exceed 6-10 GHz or more. We probably want to keep our grounds solid and low impedance over all the useful bandwidths of our circuits or else we are liable to build an oscillator by mistake. This can happen if some part of our circuit (even the ground planes) resonates in the amplifier’s useful gain bandwidth with enough gain and with high enough Q to cause an oscillation.

A second caveat: This spacing principle may not apply to ground vias used in power circuits or decoupling circuits around ICs, especially digital ICs. Remember that a digital signal has to pass at least the 5th harmonic with decent fidelity to keep those square waves really square.

**Measurements**

To prove this to myself I made a simple coplanar over waveguide PCB sample [1] and spaced the vias at 200-mil intervals along the trace. Then I ran a S21 through-line measurement with my network analyzer (Figure 3).
Theoretically the 200-mil spacing would be about 1/8th of a wavelength at 4 GHz. Sure enough some slight deviation in the S21 response can be see at about 5 GHz, but things don't really get out of hand until about 9 GHz where a rather large dip occurs; past this the response really falls apart. This is all because the ground plane sections between the ground vias no longer look like a low impedance ground, but are actually resonating.

**Other Spacing Considerations**

Sometimes via fences are built to isolate sections of one circuit from another. This is especially useful in RF and analog design where one section of the circuit must be isolated from another.

Spacing the ground vias at 1/8th of a wavelength gives good isolation here also, but for the ultimate isolation available (i.e., greater than the 120 dB that a modern network analyzer can measure) it has been found that the via spacing should be kept to 1/20th of a wavelength or less. The design problem here is really one of a “waveguide beyond cutoff” type of a design problem and has been analyzed that way in the past [7][8]. The results of these sorts of analyses compare favorably with the 1/20th-of-a-wavelength rule of thumb and my experience in designing very-high-isolation switch matrix PCBs.

![Figure 3: A sample coplanar over waveguide PCB was built according to reference 1. The via spacing was set to 200 mils, this corresponds to a 1/8th wavelength spacing of about 4 GHz. As can be seen the first sign of a nice straight line loss deviation occurs at about 5 GHz. At 9 GHz a rather large deviation occurs and the response really goes really wrong after this.](image)

**Don’t worry, be happy**

The bottom line here is that the ground via spacing for most types of digital and analog circuits is pretty lax. On even high-performance circuits 100-mil spacing will operate reliably well past 6 GHz.

The other thing that I find PCB designers losing sleep over is: There are always sections of a design where you just can’t squeeze in ground vias at the interval that you would like. Since most designers are putting more vias than absolutely necessary anyway, missing one or two won’t typically make
any difference. So just do the best you can and: Don't worry, be happy...

References:
[3] In RF design, stubs are used to make reactive elements like capacitors and inductors to tune parts of a circuit, especially in filter design and matching circuits.
[5] For high-performance RF and digital work we need to keep our stubs to an absolute minimum or we end up with very reflective transmission lines, which is considered bad.