Department of Engineering

Lecture 25: Design Tricks for RF PCBS

Matthew Spencer
Harvey Mudd College
E157 – Radio Frequency Circuit Design

Department of Engineering

Stackup, Substrates and the Virtues of Ground Planes

Matthew Spencer

Harvey Mudd College

E157 – Radio Frequency Circuit Design

2

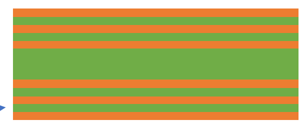
In this video we are going to discuss the vertical construction of PCBs, which has a serious effect on the performance of RF signals.

Department of Engineering

Stackups are Crucial for RF Performance

Layer order affects signal integrity

Top copper = RF
Layer 2 = ground
Layer 3 = …



Dimensions affect transmission line properties

Substrate often limits high frequency performance
- FR4 works until ~2-10GHz
- Rogers decent after that
- Fab house matters a lot

3

The term stackup refers to the design of the vertical direction of a PCB: how far apart layers are from one another, what materials they're made of, and what types of signals are predominantly routed on each layer. This has a bunch of possible consequences.

First, the order of layers matters a lot for ensuring that signals don't radiate or interfere with one another. The safest way to route a board is to ensure that high frequency signals are adjacent to ground planes, as shown in the figure, which has a bunch of benefits that we'll discuss on the next slide. Other layer tricks include placing power planes adjacent to ground planes for enhanced decoupling capacitance, and putting low frequency control signals on non-critical layers.

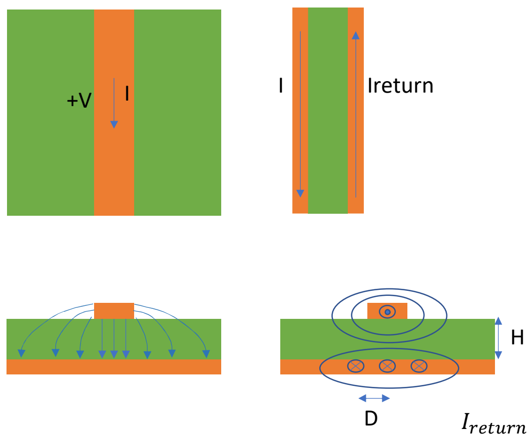
Second, the substrate material in a PCB stackup affects the performance. The substrate determine the permittivity of the PCB, and it also has a loss tangent that determines the high frequency attenuation of transmission lines on the PCB. Most PCBs are made out of a fiberglass material called FR4, and a thick piece of FR4 provides the core of a PCB. Copper sheets are then laminated onto the FR4 and etched, then thinner layers of FR4 are applied to the outside of the copper layers to repeat the process if more layers are desired. However, you can also use ceramic substrate like Rogers that have much lower loss tangents. Unfortunately, these ceramic substrates are more fragile, often toxic and, accordingly, much more expensive to use in manufacturing. That means there's a huge incentive to make boards out of FR4, but FR4 has a speed limit of about 10GHz due to attenuation. Even reaching 10GHz is quite challenging and it depends on the type of FR4 and the talents of the fabrication house. Similarly, Rogers from low-quality fabricators might underperform FR4 from high quality fabricators.

Third, the separation between the planes has a dramatic effect on the capacitance between layers and the inductance of traces. That means designing transmission lines depends on the separation between layers and even the thickness of the copper. You'll need to study these properties closely.

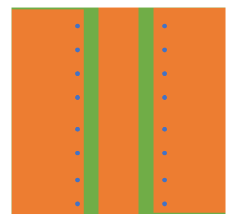
Tight field coupling, shielding. Crossovers are really bad!

HARVEY MUDD COLLEGE
Department of Engineering

Ground Planes Help Confine Fields



$$I_{return}(D) \propto \frac{1}{1+\left(\frac{D}{H}\right)^2}$$



- Ground pours are "pour" substitute
- Via fences are popular

4

It's worth expounding further on the virtues of ground planes. We can see one in action by looking at the renderings of the fields underneath the trace that's running vertically in the figure on the left. Electric field is purely vertical between the trace and the ground plane under the trace, but fringing fields cause the signal in the line to spread a little bit to the left and right. Bringing the ground plane closer will reduce the spread of the electric field. Similarly, B field loops around the current in the outgoing trace, but current in the return bath spreads out on the ground plane below the trace. Moving the trace closer pulls the return current distribution in closer.

One the right I've drawn a substitute that is sometimes used for a ground plane called a ground pour. Ground pours are large sheets of metal on the same layer as a trace that are connected to ground. They don't work as well as ground planes because fields need to couple to the left and the right. I wouldn't recommend relying on them without an underlying ground plane except in extreme circumstances like two-layer boards.

I've also drawn a bunch of blue dots on the ground pour to indicate vias. This pattern of vias is called a via fence, and you use it when you want to connect a ground pour to an underlying ground plane. RF deisgners I've met are very excited about via fences, arguing that they help with signal shielding, but I think they mostly serve a cosmetic purpose. Via fences do an admirable job of connecting pieces of metal on the signal layer to the ground plane, which is a worthy end in itself. You don't want big floating pieces of metal near your circuit, via fences can help with that. (Though be careful, via fences can make accidental antennas if you're not careful.)

It's worth noting that ground pours are important if you're trying to make coplanar waveguides, which are a specific kind of transmission line where field deliberately terminates to the left and right, and that via fences are important to ensure that your coplanar waveguide is well grounded. You do that by reducing the spacing between your trace and your ground pour. However, if the spacing between the trace and the pour is 5H or greater, then the ground pour doesn't really interact with the trace electrically.

Department of Engineering

Summary

- Stackups specify materials, thicknesses and purposes of layers in PCBs
- Stackups affect RF performance.
- Substrates limit high frequency b/c of loss tangent, FR4 max ~10GHz.
- Ground planes should be near fast signals. They confine fields.

5

Also ASK vs. AM, FSK vs. FM, PSK vs. PM

Department of Engineering

Trace Width, Routing Corners and Coupling

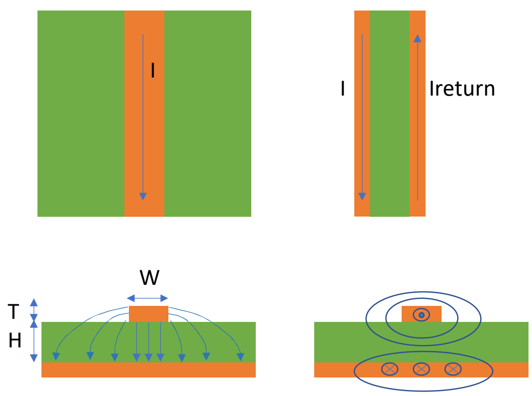
Matthew Spencer
Harvey Mudd College
E157 – Radio Frequency Circuit Design

6

In this video we’re going to talk about how to make transmission lines on an RF PCB, and about some design considerations for those transmission lines.

Department of Engineering

Trace Width Determines T Line Impedance



Microstrip Impedance vs. Width

$$Z_0 = \sqrt{\frac{\mu_r \, \mu_0}{\epsilon_r \, \epsilon_0}} \frac{H}{W_{eff}} \left[1 + 1.735\epsilon_r^{-0.0724} \left(\frac{W_{eff}}{H}\right)^{-0.836}\right]$$

with $$W_{eff} = W + \frac{T}{\pi}\left[\ln\left(\frac{2H}{T}\right) + 1\right]$$

7

The stackup of a PCB is at the heart of designing a transmission line because the separation between layers trades off between capacitance and inductance per unit length. The inductance per unit length is proportional to the amount of flux coupled in a current loop, which is formed by the current in the trace and return current in the ground plane. Wider separation allows more field between these currents. In contrast, wider separation results in weaker electric fields between the plates.

Similarly, trace width affects both capacitance and inductance. Wider traces mean more electric field is coupled between the plates, but they make the magnetic field around any loop of current weaker.

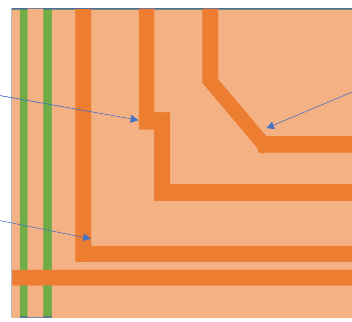
These effects are captured in the equations for characteristic impedance of a microstrip equation shown on the right. The ratio of trace width to stackup separation height appears prominently in them. This is one equation for microstrip impedance out of many because lots of approximations have been developed over the years. My usual approach to sorting through all these equations when designing a microstrip is to poll all of the online microstrip calculators and pick the mean of their slightly varying answers.

Microstrips aren't' the only transmission lines you can build on a board, and you should take the time to familiarize yourself with the profiles and impedance formulas for striplines and coplanar waveguides, both of which can be useful tools in your belt.

While we've talked a lot on this slide about how to make a transmsission line, but not really about why you would. Certainly, you could use transmission line to connect RF components, which is most of what we've talked about in this class. Transmission lines are also sometimes useful for high speed digital signals because high frequency content in the sharp edges of digital signals can reflect off of unterminated receivers. Don't forget that you need to include termination resistors, often explicit ones that you add to logic gates, if you're going to use transmission lines for high speed digital. There's a lot of debate in the community about whether it's more important to have your source or your load match the transmission line impedance, but it's safest to have both source and load match.

Department of Engineering

Corners Don’t Matter, Proximity Can Matter



8

Extremely strong aesthetic preferences are common in the radio frequency circuit board design community. You’re going to be told often that your wires need to be straight and that you should never make a ninety degree turn with a high frequency wire. I recommend you follow that advice because following them produces nice looking boards, forces you to think carefully about floor planning, and keeps your strident co-workers happy. However, most of these design practices are myths: they won’t have a significant effect unless your input signal is in the 10s of GHz range.

To be specific about some of these preferences, one classic argument is that 90 degree corners in transmission lines have a little bit of extra capacitance because the fringing fields from the vertical and horizontal wires add up in the corner. This is true, but it only causes a minor change in line impedance unless your wavelength is similar to the width of the trace. At those frequencies, skin effect, which is a phenomenon where current crowds to the edge of traces at high frequencies, will probably swamp out this minor change in capacitance. Because corners are frowned on, jogs are doubly condemned because they have lots of corners. However, if no corner causes a significant problem, then lots of them aren’t a big deal. As a rule of thumb, a wave will ignore a feature that is electrically small, which is to say much smaller than its wavelength or the speed of light times its rise time. Chamfered corners are often recommended as fixes to ninety degree corners.

On the other hand, putting traces too close together is dangerous because signals can couple from one another. Signals with fast changes in voltage can capacitively couple to adjacent traces or especially to traces on another layer, CLICK, like this one. In addition, traces carrying high current can coupled field through other nearby traces, which will induce current in them through mutual inductance. Pay careful attention to trace spacing, especially on high power or fast-switching traces.

One final sin that I've committed in this board layout is crossing a top layer trace over a gap in the ground place. In addition to coupling to the underlying trace, this has the result of breaking up the current return path for the trace on the top layer. That adds a lot of parasitic inductance to the trace because of the bigger loop the return current travels through, and it also turns the underlying break int the ground plane into a slot antenna that radiates when we drive the top layer. In general, breaks in the ground plane are very dangerous and should only be used if you know what you're doing.

Department of Engineering

Summary

- Trace width trades off inductance and capacitance per unit length, which means it determines Z0

- Corners don't matter in traces, but coupling and multi-layer crossovers do.

9

Department of Engineering

Components and Connectors

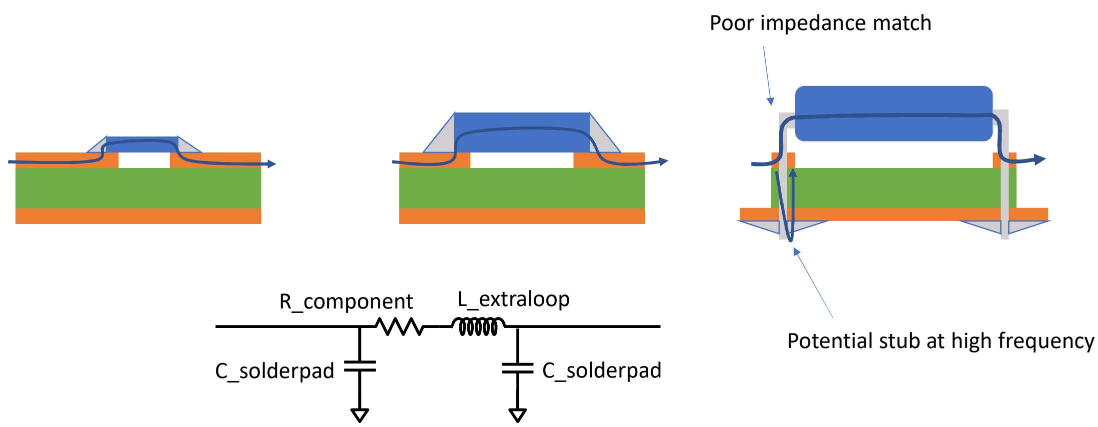
Matthew Spencer
Harvey Mudd College
E157 – Radio Frequency Circuit Design

10

In this video we’re going to talk about practices for picking components and designing traces that interface with them.

Department of Engineering

Components Have Parasitics



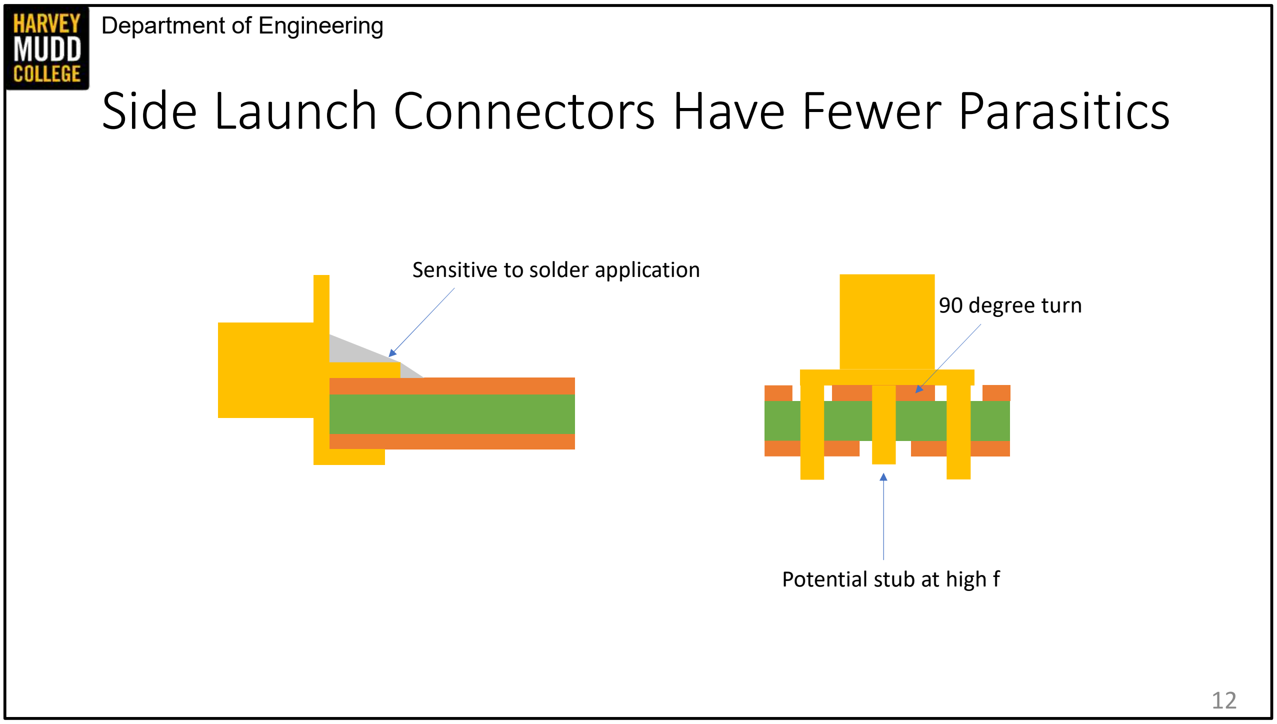
11

Components all have parasitics, but they are especially pronounces at high frequency. Here I've drawn three resistors, two in surface mount packages and one in a thru-hole package. I've also included a model for a high frequency resistor. The silver triangles outlined in blue indicate solder connections.

The current has to move farther from the ground plane as it moves through the component, which introduces some inductance. The pads, which are often wider than the traces that connect to them, introduce capacitance to lower layers, and the solder can introduce lots of stray capacitance. Even worse, if the solder isn't connected evenly across the component, then there can be constrictions of current that introduce inductance or contact resistance.

Taken together, all of these effects suggest that smaller components have fewer parasitics: the inductive loops are smaller, less solder is used, and the contact pads are smaller, which reduces stray capacitance and makes them easier to interface with narrow transmission lines. Small components are also more likely to be smaller than the wavelength of a signal, which makes it easier to model them in circuit analysis. Finally, small components are also easier to assemble (for robots, not undergraduates) and take up less board area, so many passives used in RF designs are in tiny 10x20 mil packages.

CLICK Thru-hole components are especially bad for RF designs. They're notorious sources of radiation and coupling because currents in thru-hole components can be very far from the ground plane, because the wires that poke through the circuit board can create reflections at very high frequencies, and because the wire leads are often poorly controlled impedances that look little like transmission lines. The wire leads can be rich sources of radiation too if you're unlucky in your layout. Finally, they're simply bigger, which means that you need to consider them as transmission lines at lower frequencies. Don't use thru-hole components for RF designs.

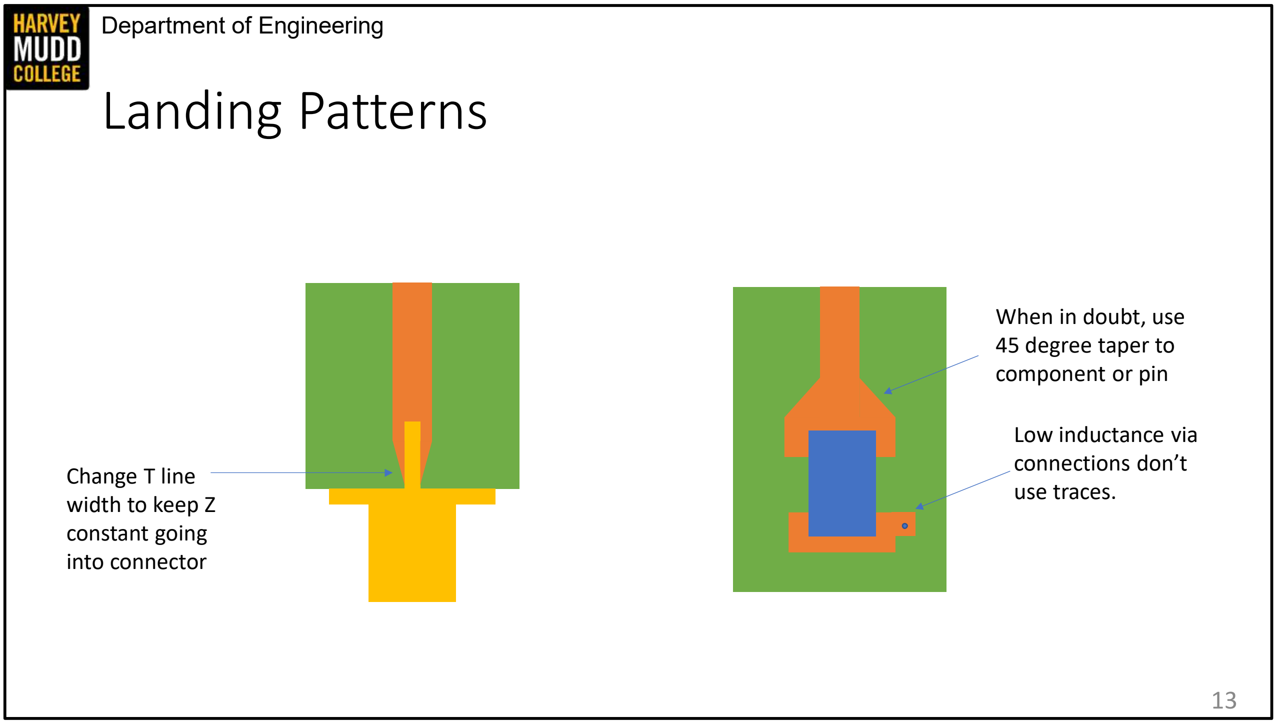


Connectors also matter a great deal in the design of RF systems. They're very common failure points because they are places where a signal passes from one transmission line to another, which can cause reflections. As a result, your efforts as a designer and the quality of assembly is crucial for connectors.

Connectors come in larger and smaller sizes, and small connectors like UFL are popular for the same reasons that small components are popular. That said, SMA connectors are the workhorse for lots of mid-GHz designs. Designers tend to switch to UFL for very compact designs and 3.5mm for very high frequency designs.

One design decision that you have to make with connectors, particularly SMA connectors, is whether the connector will launch from the side of the board of the top of the board, which are the two figures illustrated on this slide. Side launch connectors present a smoother transition from a wire to an on-board trace because the signal travels in a straight line. Vertical connectors force the signal to turn ninety degrees, which is a bigger deal here than with traces because of the many parasitics built into connectors. As a result, side-launch connectors are preferred when possible That said, connector designers know their business and following layout recommendations for a vertical connector will usually result in a successful connection. That's good, because the number of side launch connectors on a board is limited by the perimeter, and it can be difficult to route from the interior of the board to the edge. Don't be afraid of a vertical connector when appropriate, but prefer side launches.

One other wrinkle for assembling vertical connectors is that lots of them are thru-hole, which means they can introduce stubs like thru-hole resistors. Further, the crucial solder joint that determines if the signal pin mates well with the on-board trace is hidden under the connector.   Side-launch connectors are also sensitive to the solder job on that connector, but you can see inspect the solder job easily.



Finally, your transmission line is unlikely to be the same width as your components and pins, especially if they have extra pads, which presents some challenges for routing to them. A step change in width represents a sudden change in line impedance that can cause reflections. Tapers or expansion of line width to mate with pins and components are called landing patterns and you can usually get them from component designers, especially from connector designers, who will sometimes assign you an application engineer to make a custom pattern for you.

However, if you can't get a manufacturer to bother with giving you a landing pattern, which will be the case for lots of passive components, then a 45 degree taper often does an OK job, especially if the pad dimensions aren't too far from the line dimensions. That taper is a weird piece of metal, so I often build it into the footprint of the part in my PCB layout program rather than trying to modify each trace that connects to a similar part.

Finally, it's worth noting that vias are quite sensitive to layout. Additional trace length that connects to vias can introduce inductance, which can have lots of undesired effects. For instance, I've see poor via design introduce instability in amplifiers if the ground isn't stable at high frequencies. Very low impedance vias will connect to directly to pads rather than being connected through wires. You can also sometimes put vias in the middle of pads, but doing that can violate design rules at some manufacturers because they affect heat sinking and solder filling properties.

Department of Engineering

Summary

- Small components have fewer parasitics than big ones, especially thru-holes. (They also match small transmission lines.)
- Side-launch connectors often perform better than vertical ones, especially thru-hole vertical connectors
- Landing patterns are crucial for performance. Get them from manufacturers or fall back on 45 degree tapers.
- Low impedance vias don't use traces.

Department of Engineering

0 Ohm Resistors and Debugging Techniques

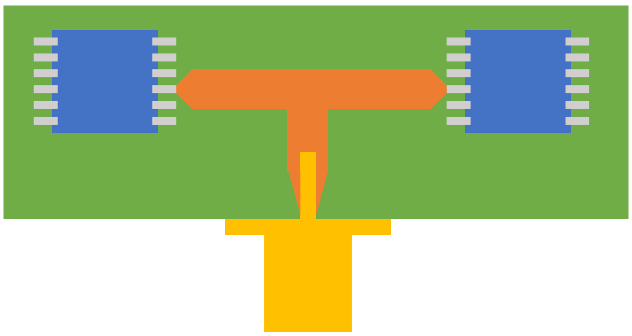
Matthew Spencer
Harvey Mudd College
E157 – Radio Frequency Circuit Design

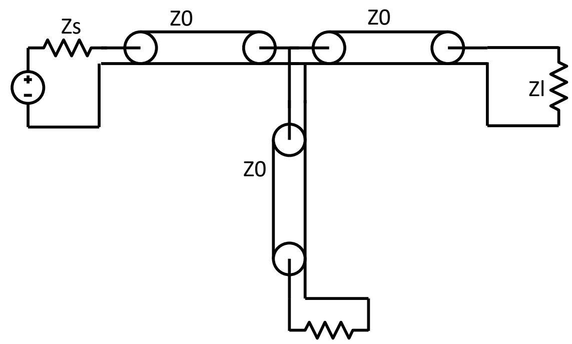
15

In this video we're going to learn about techniques to reconfigure high speed PCBs so that you can inspect signals without disrupting them.

Department of Engineering

Test Equipment Creates 50 Ohm Loading



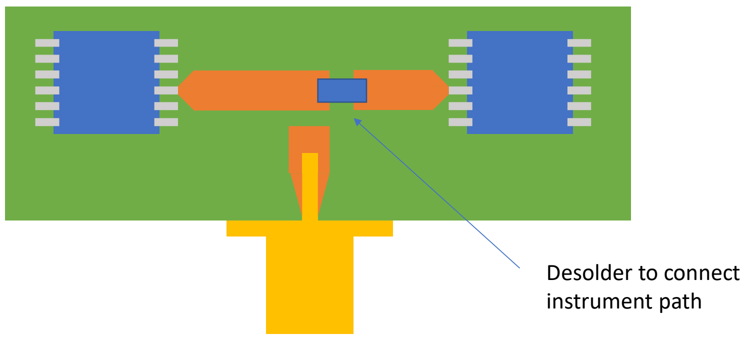


16

This video is inspired by a technique that works well at low frequencies, but causes tremendous grief at high frequencies. Here I've shown two chips attached to a transmission line, one of which is sending a signal to the other. For some reason, we think this signal is interesting and would like to inspect it off chip. However, if we make a circuit model of this situation then we see that this off-chip inspection is very problematic. The transmission line leaving the transmisster sees two transmission lines at the T-junction, which results in a large impedance mismatch. Even if we custom design the traces to act like a splitter, we still have the issue of splitting the power that comes out of the transmitter. Finally, if we disconnect our instrument so that Zinstrument becomes an open circuit, then the vertical trace will act like a high frequency stub that can introduce disruptive refections into our trace.

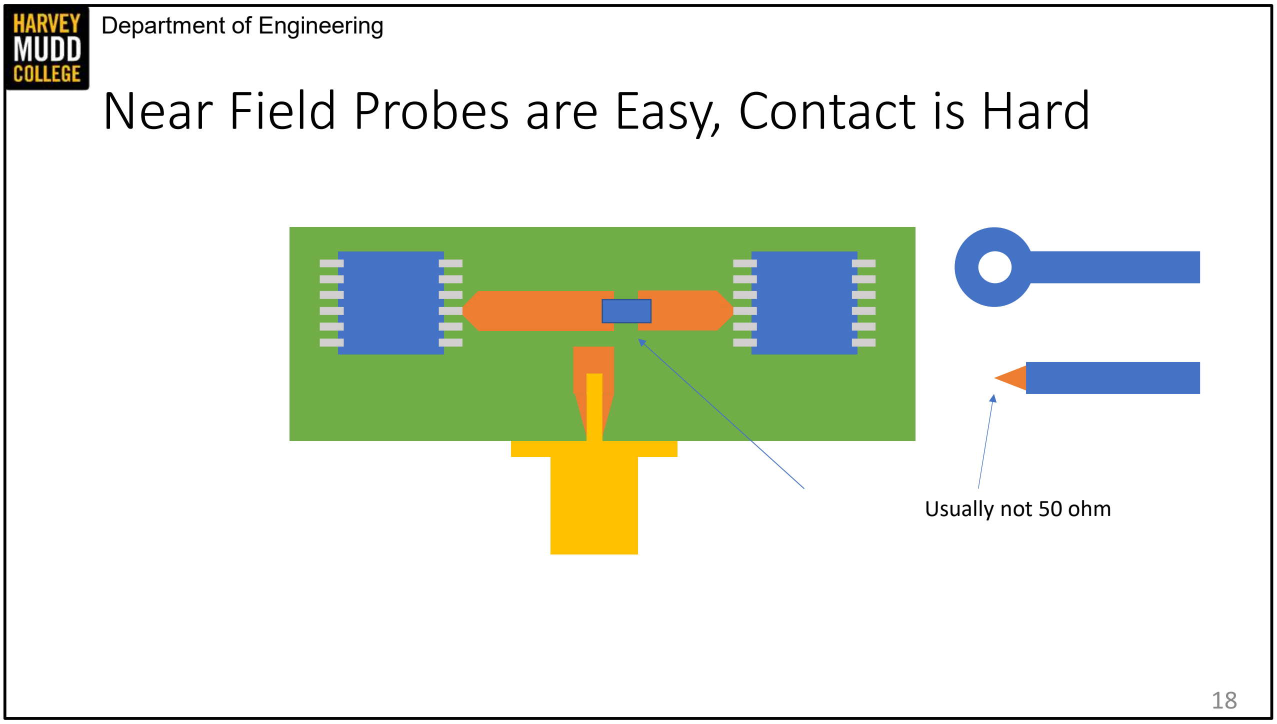
Department of Engineering

Use Zero Ohm Resistor As Jumpers



We can fix this by using components called zero ohm resistors as jumper connectors. By soldering a resistor into the forward position or the instrument position we can decide where we want our signal to go without any impedance mismatch issues. Zero ohm resistors can provide some much needed configurability to high frequency systems without the parasitics of jumper pins or discrete switches. This is good for chasing down instability in systems and all kinds of other debugging.

As an aside, you could also use a directional coupler if you needed to measure the signal with the instrument during operation.



It's good that zero ohm resistors work well, because probing RF systems is difficult. Near field probes can give us some information without contacting traces, and they are relatively easy to use, often with spectrum analyzers. However, contact probes like for oscilloscopes are difficult to use at high frequencies. They tend to have a big impedance mismatch with the system, so they pick up little RF power. That means they don't disturb your system, but also that your oscilloscope won't get much signal. If you do find a probe that is well matched to your system, which is a challenge, then you have the same problem we started with where the probe can interfere with existing transmission lines on your board.

Department of Engineering

Summary

- Connecting equipment without thinking about it can cause reflections by messing up trace impedance, adding extra loads.
- Zero ohm resistors let you configure connections w/ low parasitics.
- Near field probes are easy to use, contact probes are harder.