Further PCB Design and Manufacturing

Josh Johnson

12/8/2019

FPGA Workshop?

My proposal is:

- Two workshops
  - WTFpga - Implementing a 7 segment decoder at your own pace
  - LED Matrix / UART / Simulation
  - Verilog only (synthesis tools don't support SV/VHDL)
- Requires Linux / Mac, may work on windows.
  - Will provide untested instruction for windows which may require different workflow - pull requests welcomed
- FPGA hardware will be $50
  - iCE-40 FPGA in feather form factor
  - 7 Segment / DIP switch board
  - 6*6 LED Matrix / Buttons
  - Feather to dual PMOD (PCB only)
  - Want to assemble your own? Can be organised!
- Will get hardware to everyone before next meetup, so you can install toolchains / get to blinky at your own leisure

Overview

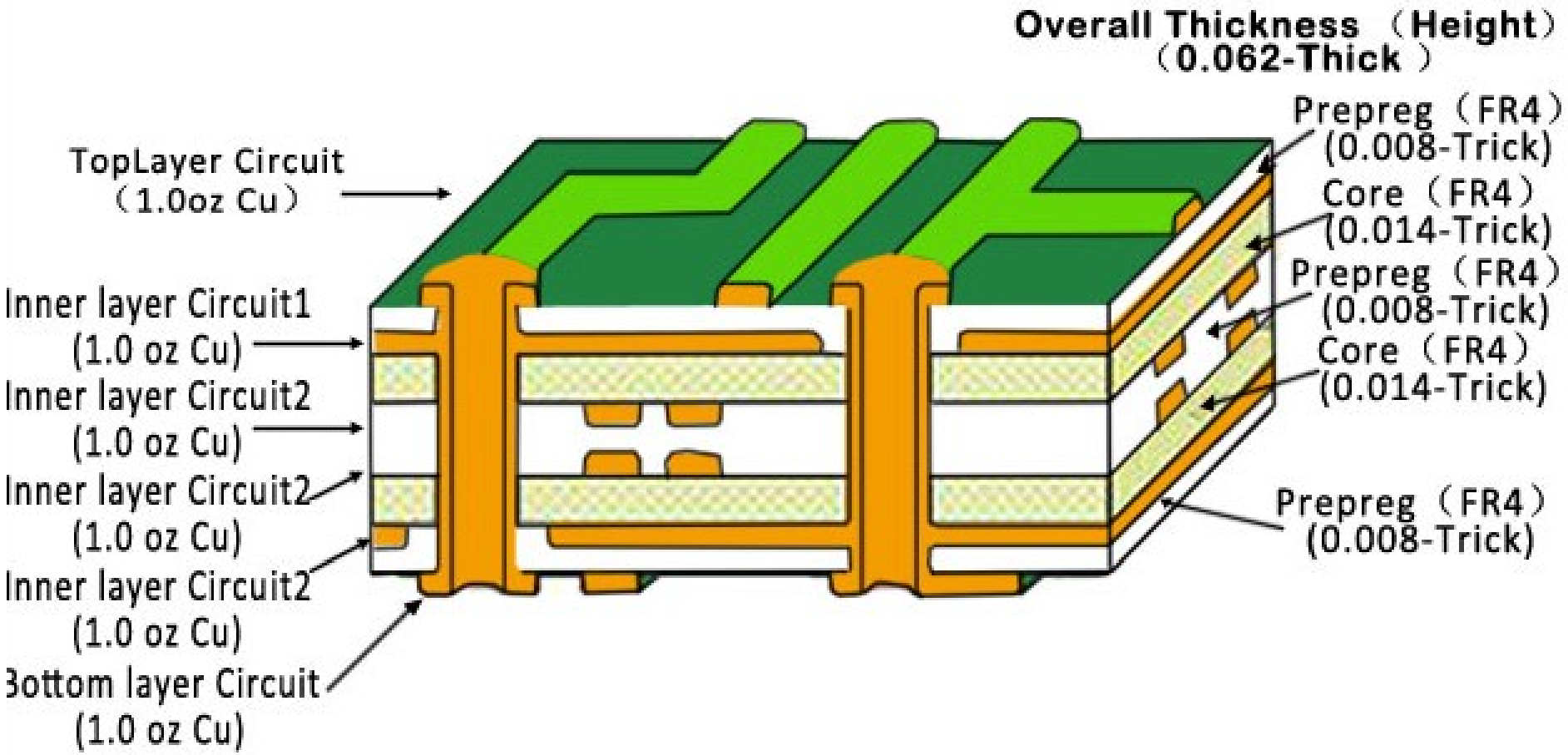
- Multilayer PCBs
- Impedance control
- Differential Pairs
- Cutting edge processes
- Panellisation
- Random KiCad things
- Layout Tips
- Layout Review

Multilayer PCBs

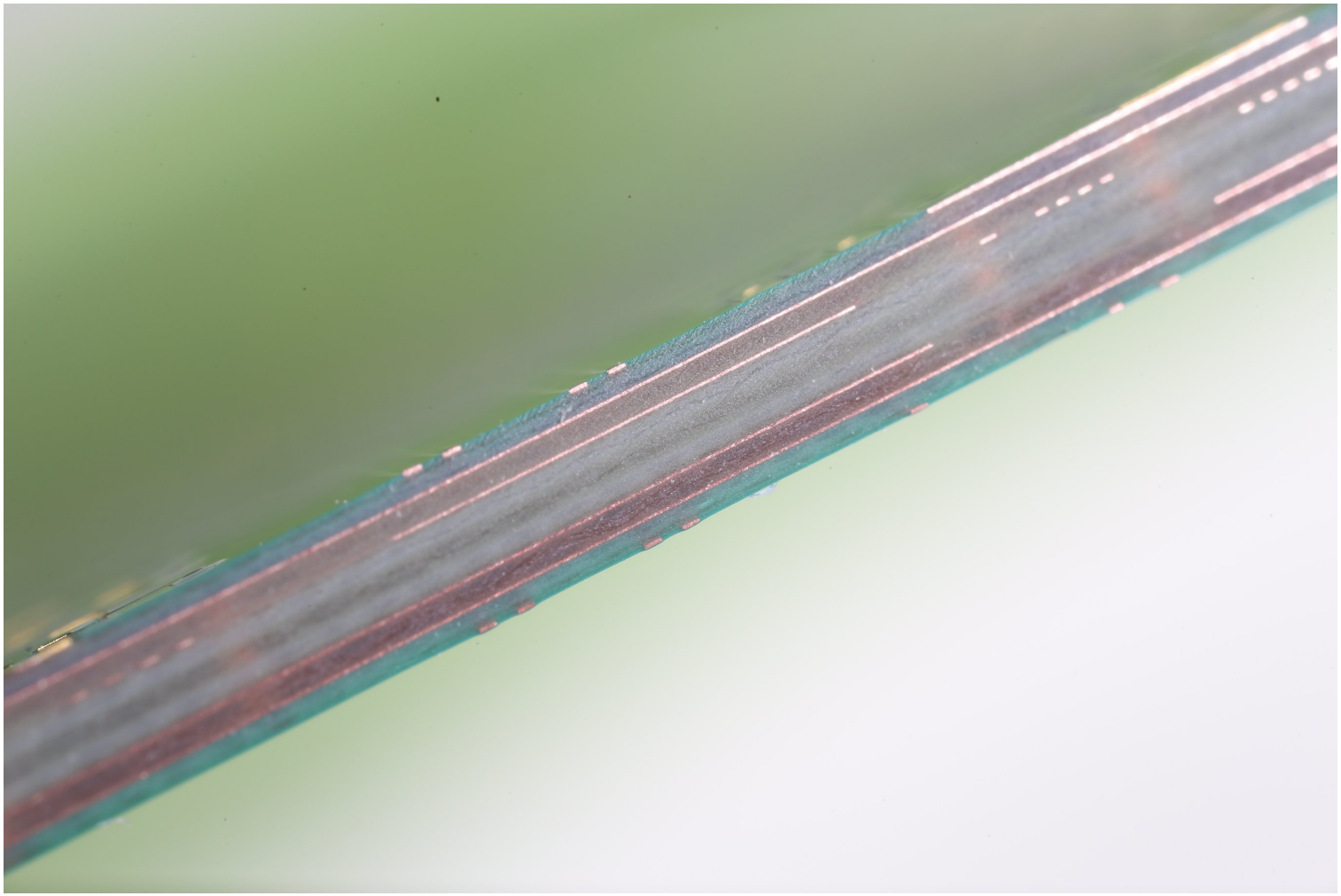
PCBs are often 2 or 4 layers, but 8-16 is common for phones / motherboards / GPUs etc. Why?

- Dedicated power / ground planes
  - Allows for low impedance supply and return of currents
  - Not only required for function of devices, but to pass emissions testing
  - Many different stackups, both material, spacing, and layer assignment (signal / gnd / power)
- More layers to route signals
- Required for BGA fanout
- Impedance control

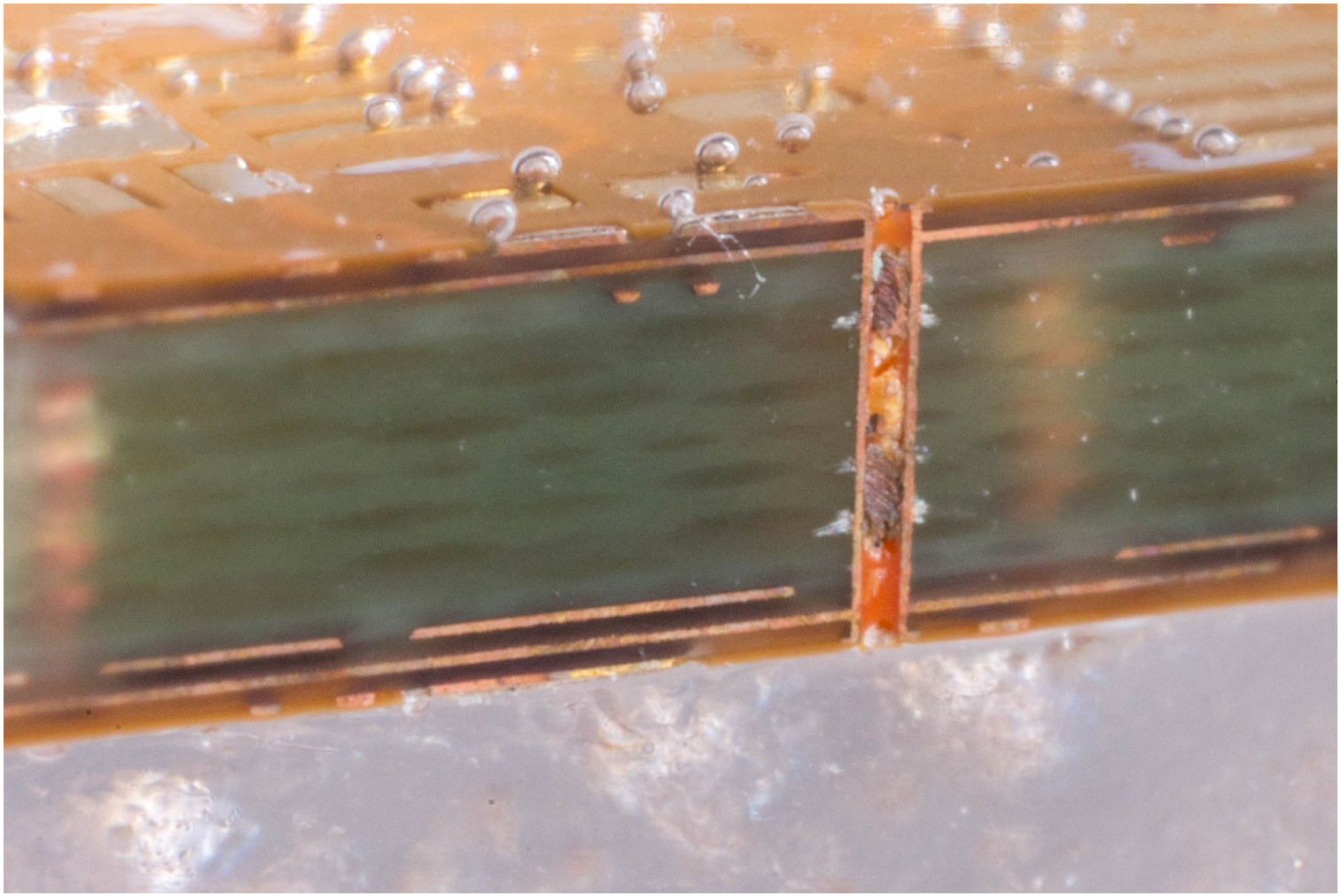
PCB Stackup


Overall Thickness （Height）
（0.062-Thick ）
Prepreg （FR4）
(0.008-Trick)
Core （FR4）
(0.014-Trick)
Prepreg （FR4）
(0.008-Trick)
Core （FR4）
(0.014-Trick)
Prepreg （FR4）
(0.008-Trick)
TopLayer Circuit
（1.0oz Cu）
Inner layer Circuit1
(1.0 oz Cu)
Inner layer Circuit2
(1.0 oz Cu)
Inner layer Circuit2
(1.0 oz Cu)
Inner layer Circuit2
(1.0 oz Cu)
Bottom layer Circuit
(1.0 oz Cu)

PCB Stackup

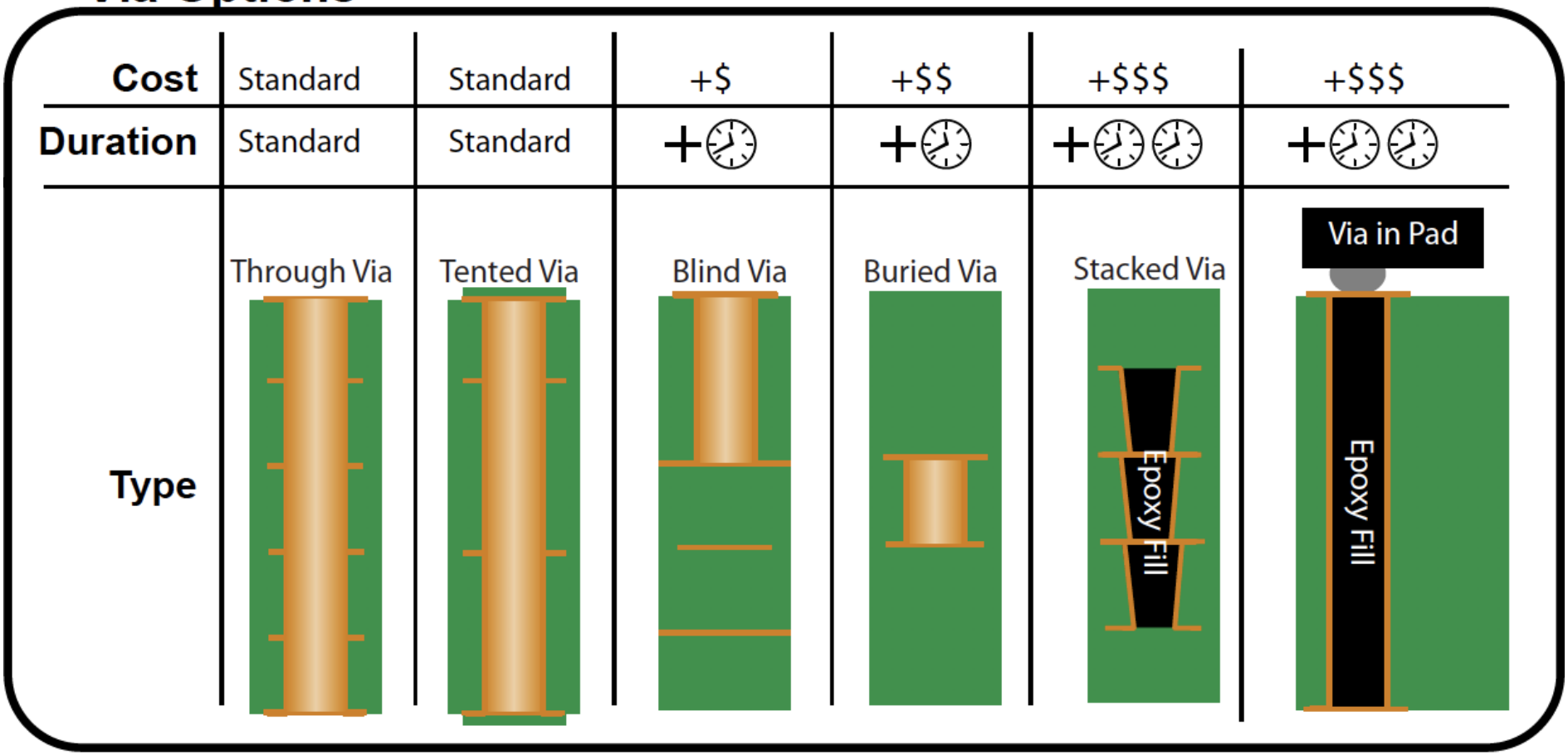


PCB Stackup



Via Technology

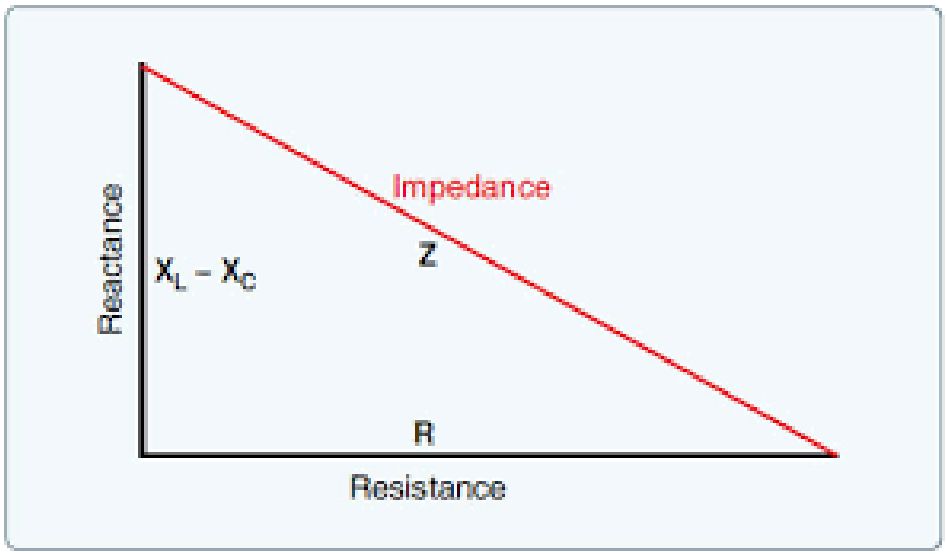
Via Options



Impedance Controlled Traces

What is impedance?

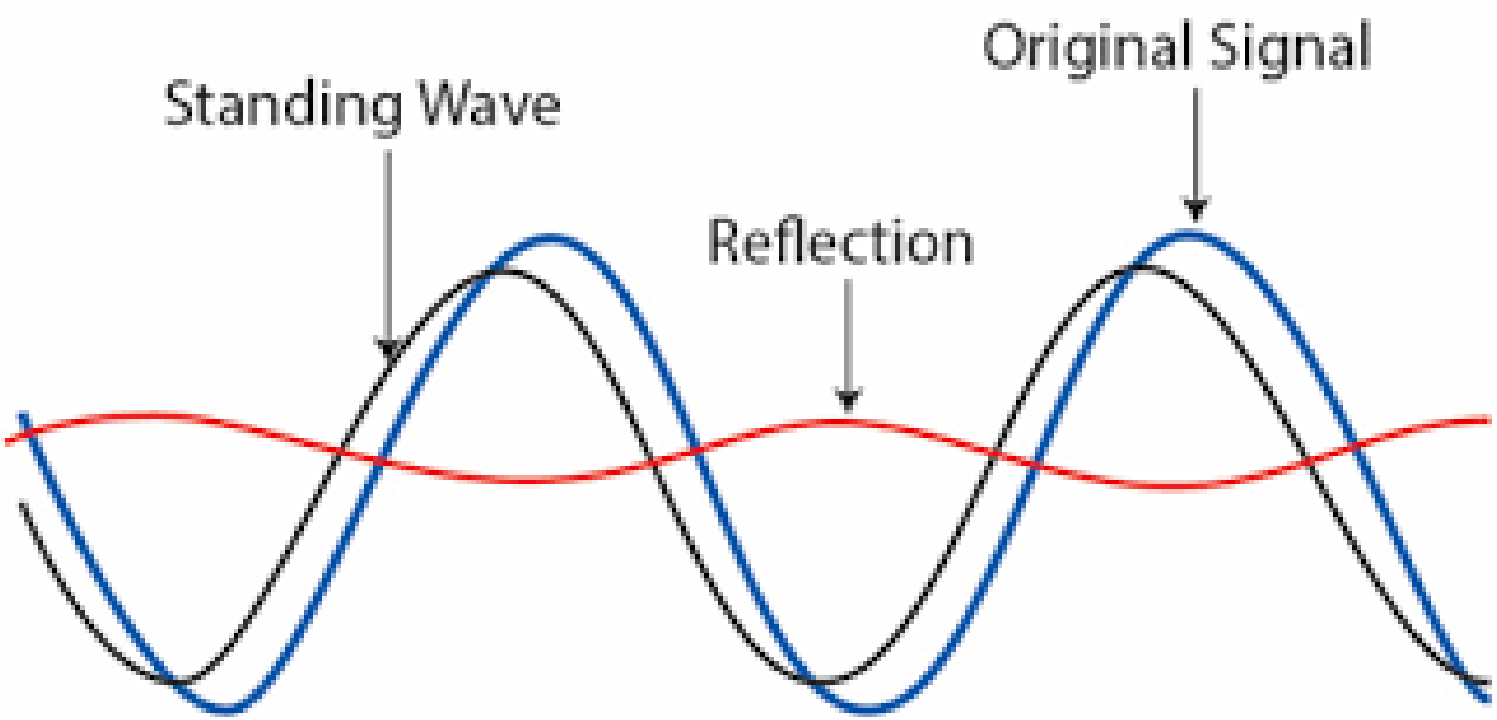
- the effective resistance of an electric circuit or component to alternating current, arising from the combined effects of ohmic resistance and reactance



Impedance Controlled Traces

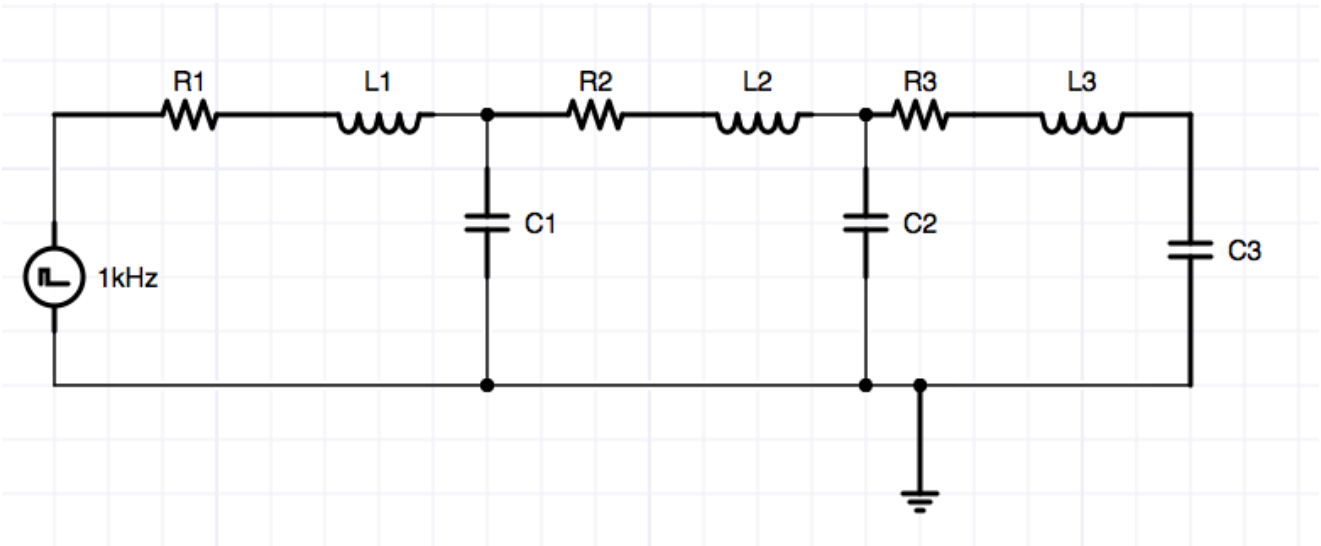
Why do we care?

- Impedance mismatch results in reflections, which causes power loss and signal degradation
- Mismatches caused by incorrect track width, stubs, lack of termination resistors



Impedance Controlled Traces

Transmission line model

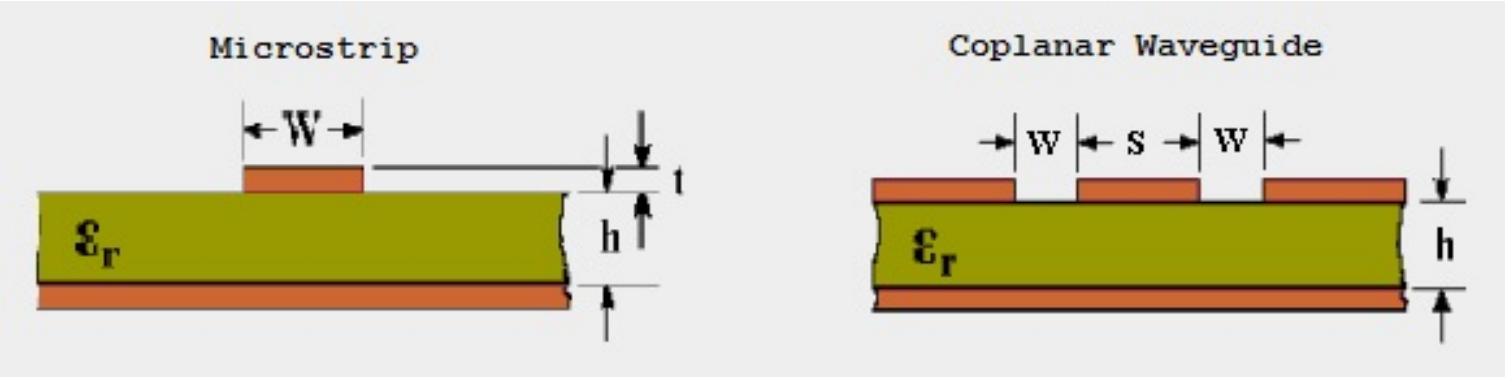


Characteristic Impedance

$$Z_0 = \sqrt{\frac{L}{C}}$$

Impedance Controlled Traces

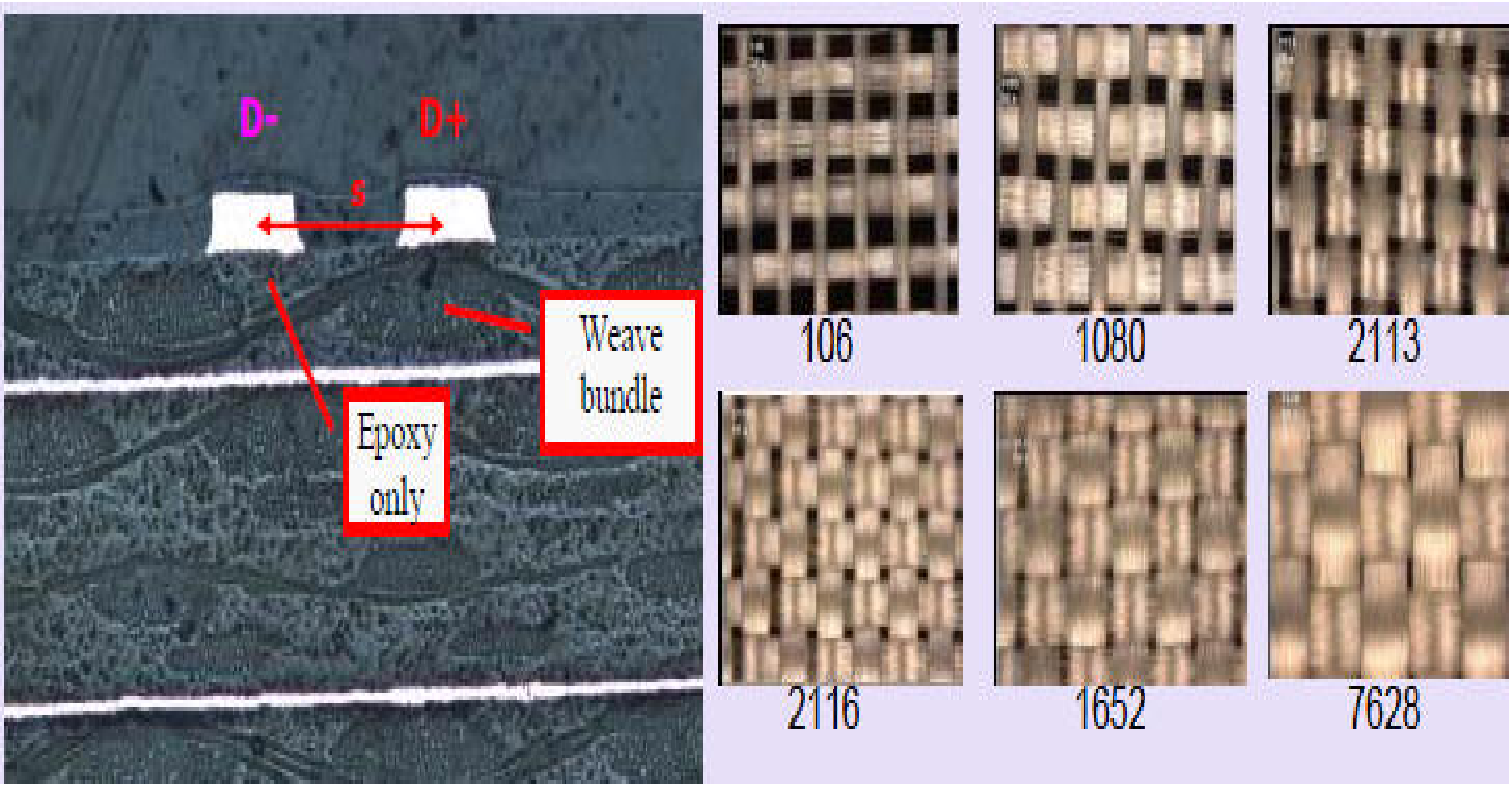
PCB transmission line



To calculate, use tools such as TXLine or Saturn PCB Toolkit

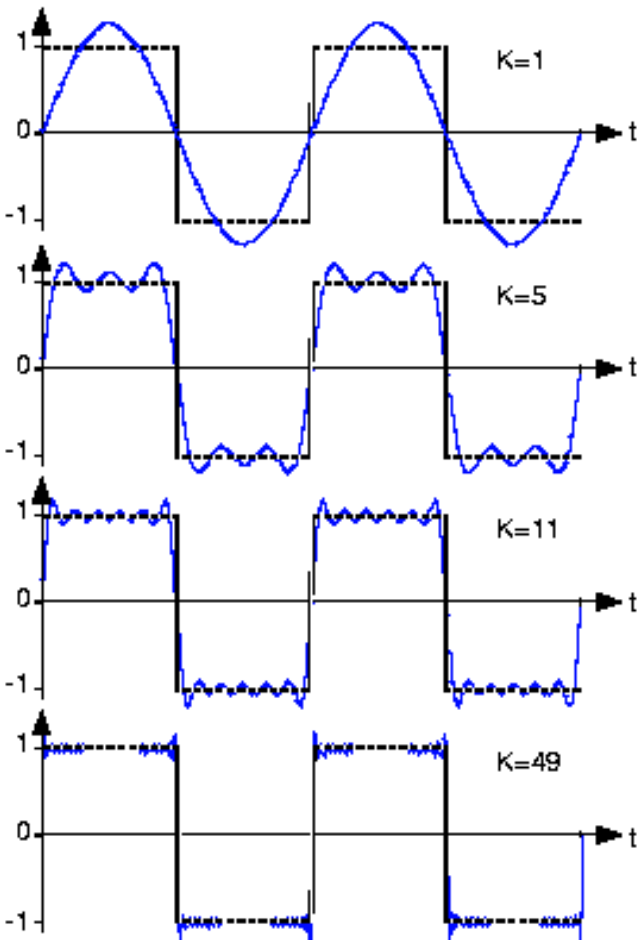
At higher frequencies, (above 2GHz) FR4 has high loss
Materials such as PTFE, Rogers 4003C / 4350B (glass reinforced hydrocarbon/ceramics) are better suited to higher frequency applications
Packing and orientation of glass weave can also play a part, along with surface roughness due to skin effect

Glass Weave



Impedance Controlled Traces

I'm designing a digital board, so I don't have to worry about this?

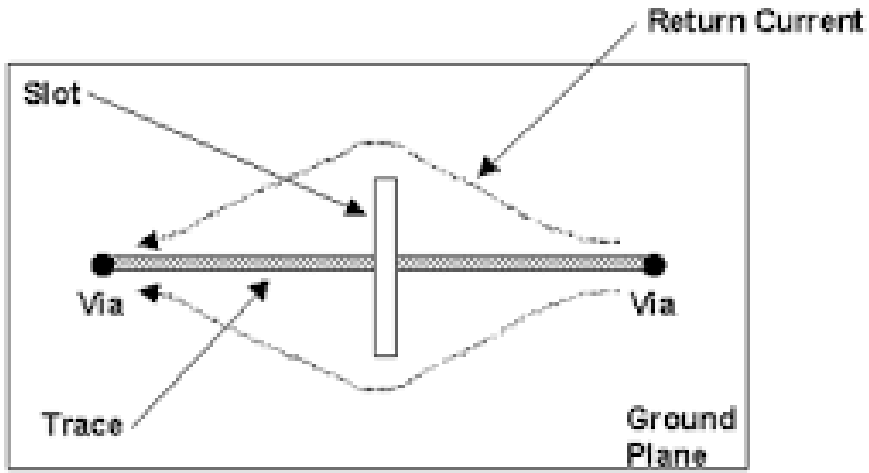


- Bandwidth $= \frac{0.34}{t_{rise}}$
- Dependent on rise / fall times, not frequency (although higher frequencies do require faster edges)

Impedance Controlled Traces

Return current?

- Impedance controlled traces must be referenced to an unbroken plane (can be ground or power)
- As return current takes path of least impedance, follows path of outbound current!
- Slots, gaps, cut outs all result in return current taking a long path and emitting EMI



SLOTTED GROUND PLANE

Path of least impedance

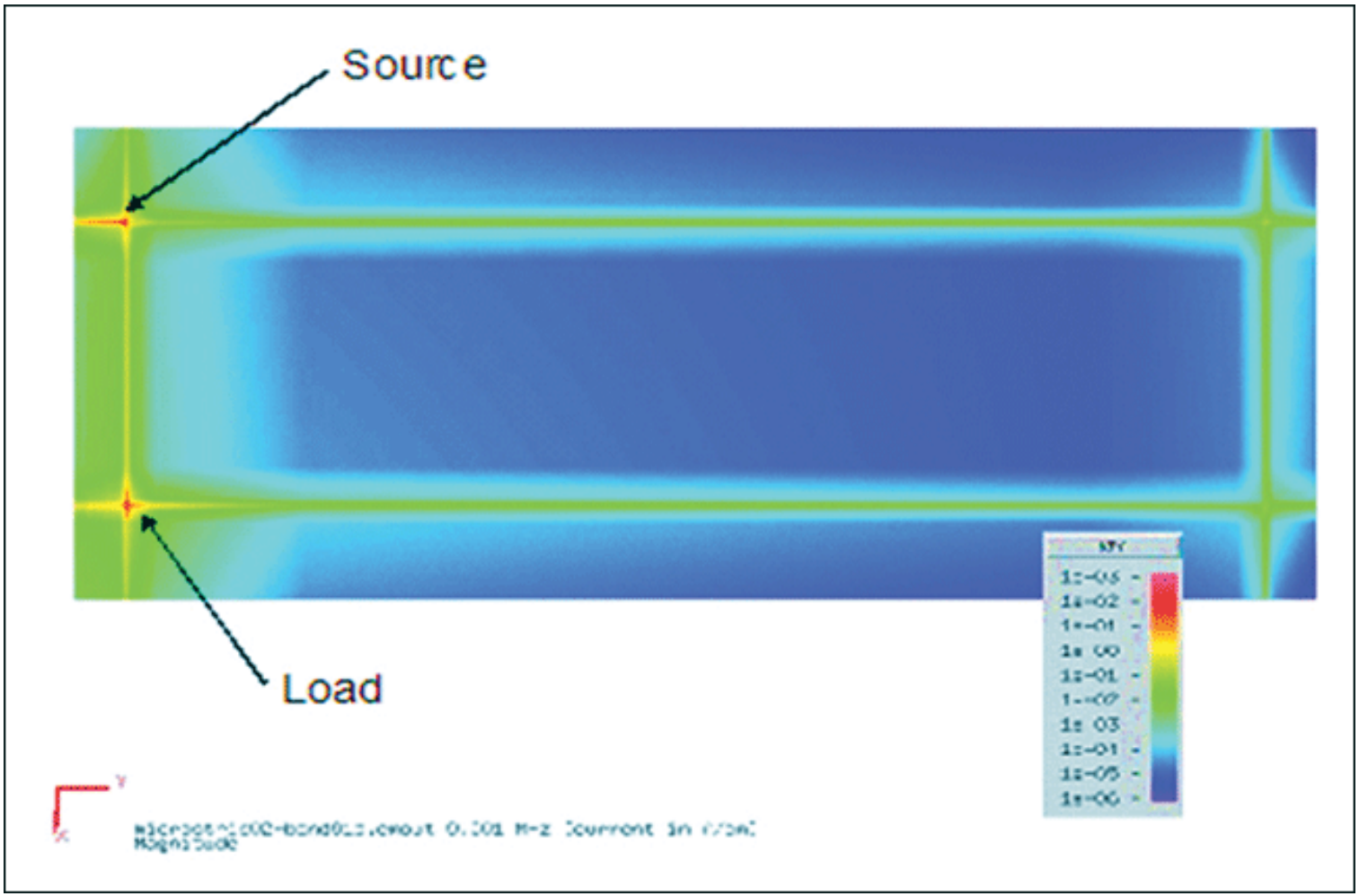


Figure: Currents for a 1kHz signal flow. The ground current primarily flows directly from the load to the source in a straight line, as indicated by the narrow yellow line.

Path of least impedance

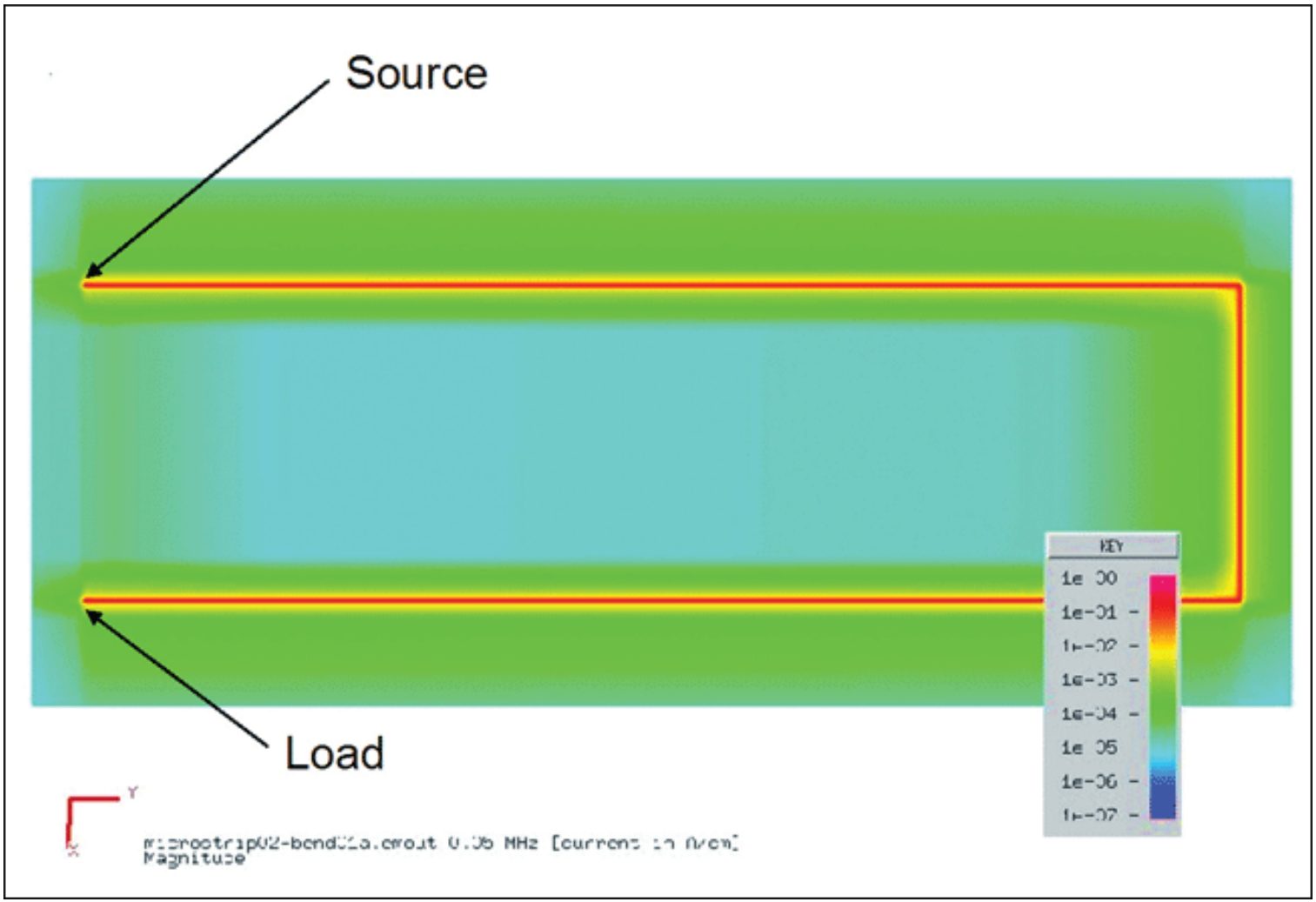


Figure: Current for a 50kHz signal flowing primarily along the signal trace and, to a lesser extent, directly from load to source.

Path of least impedance

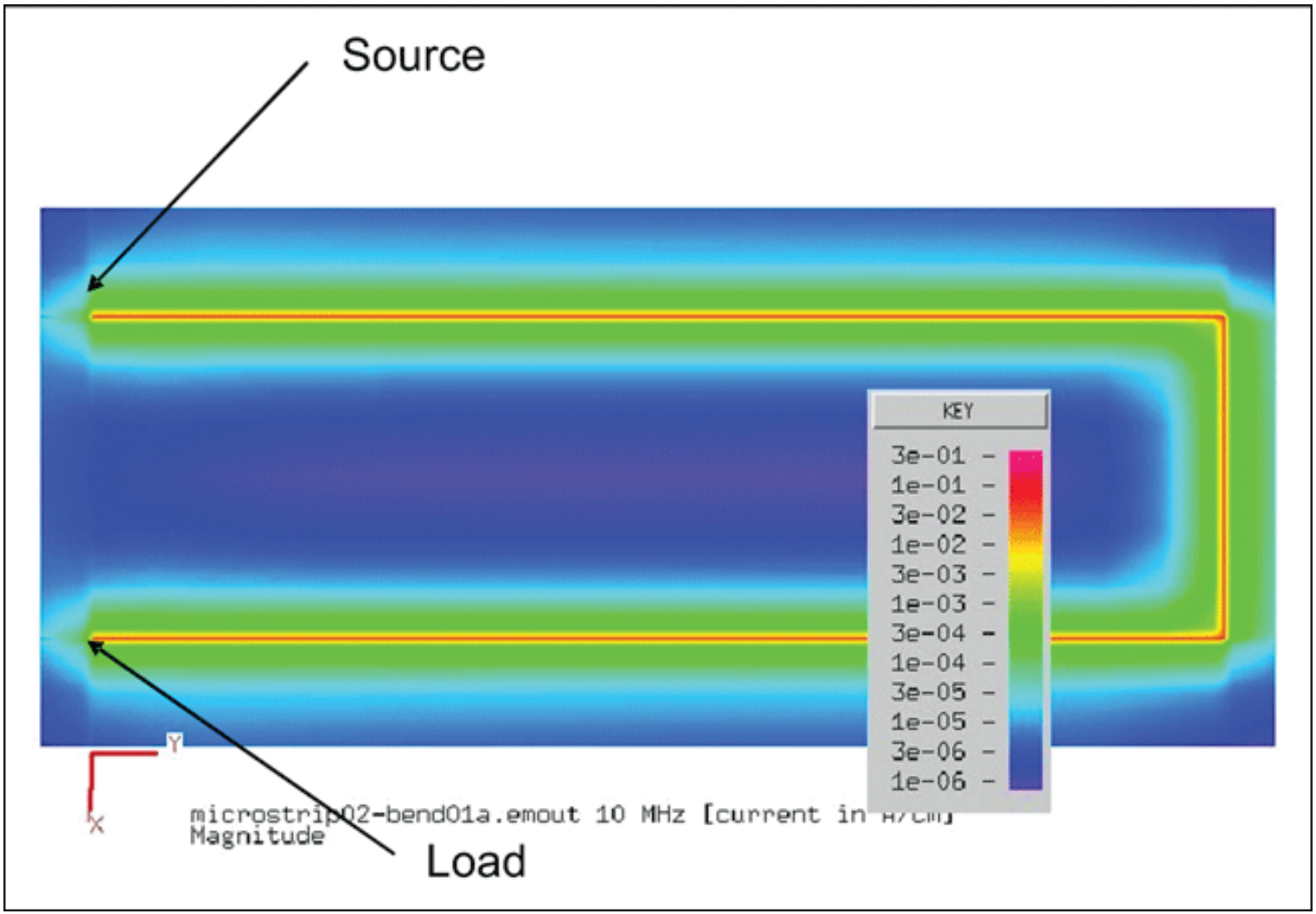
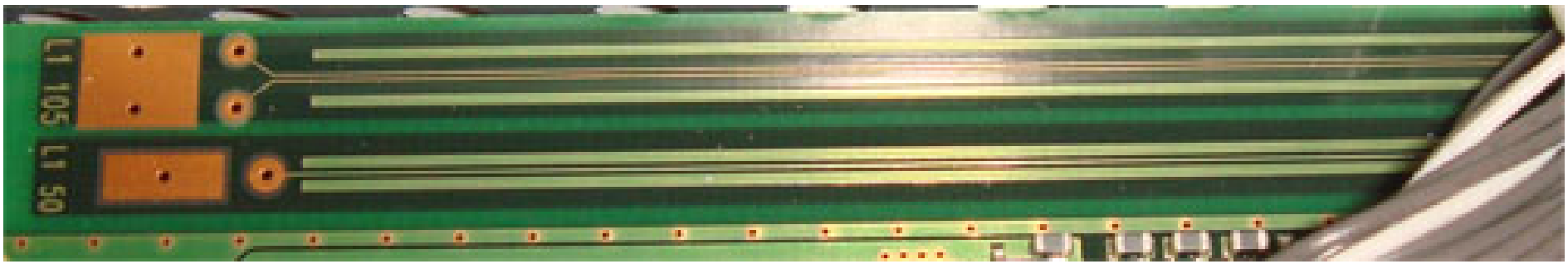


Figure: Current paths with a 1MHz signal. Virtually all the return ground current is flowing along the path of the signal trace

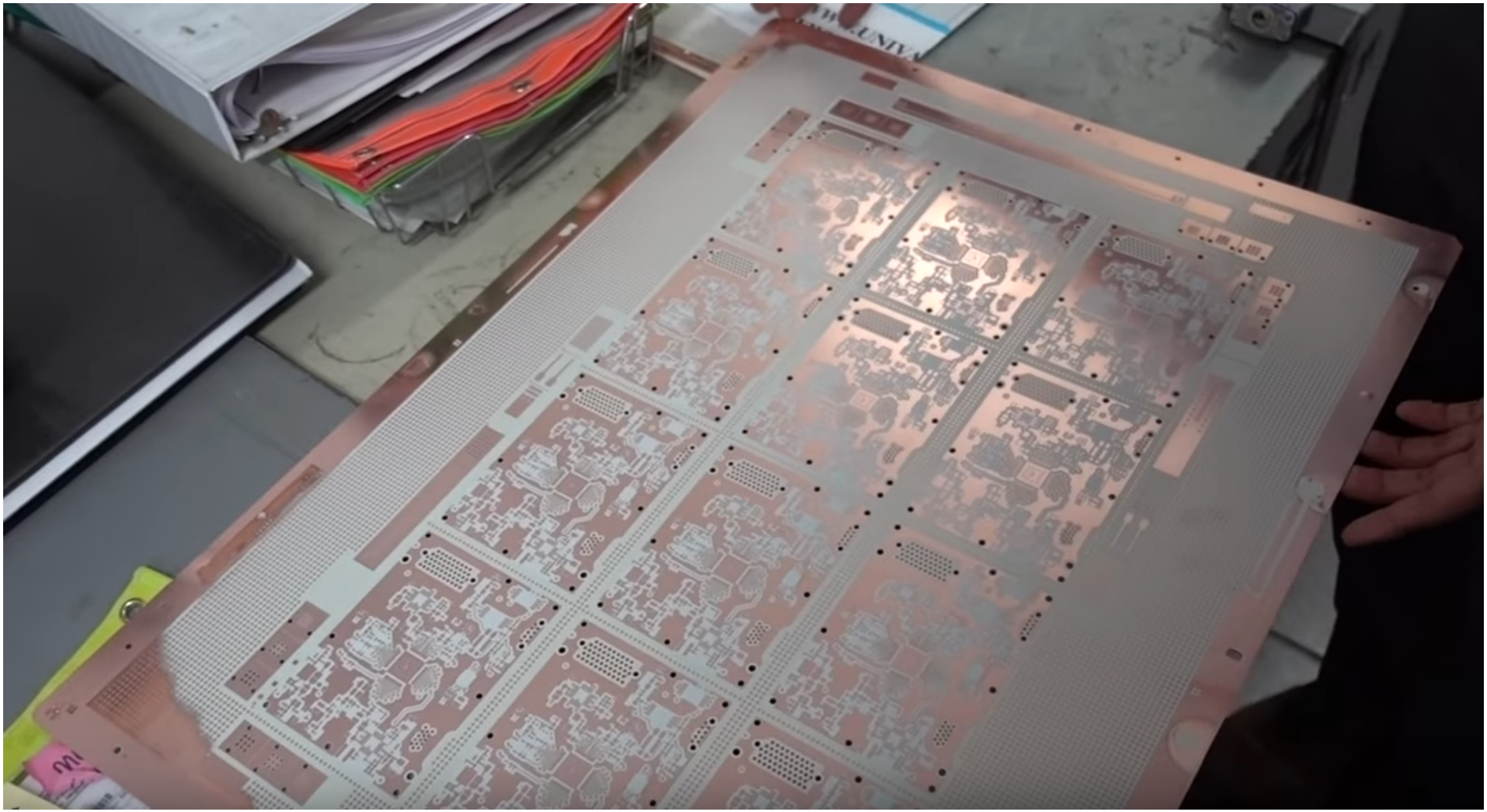
Impedance Controlled Traces

Confirming correct Impedance?

- Follow fabricators guidelines (NOT TXLine etc) they know their process best
- Board houses do what is known as 'Etch Compensation' to ensure traces stay at required impedance
- They can also make a test coupon, and measure with a TDR to ensure correct dimensions



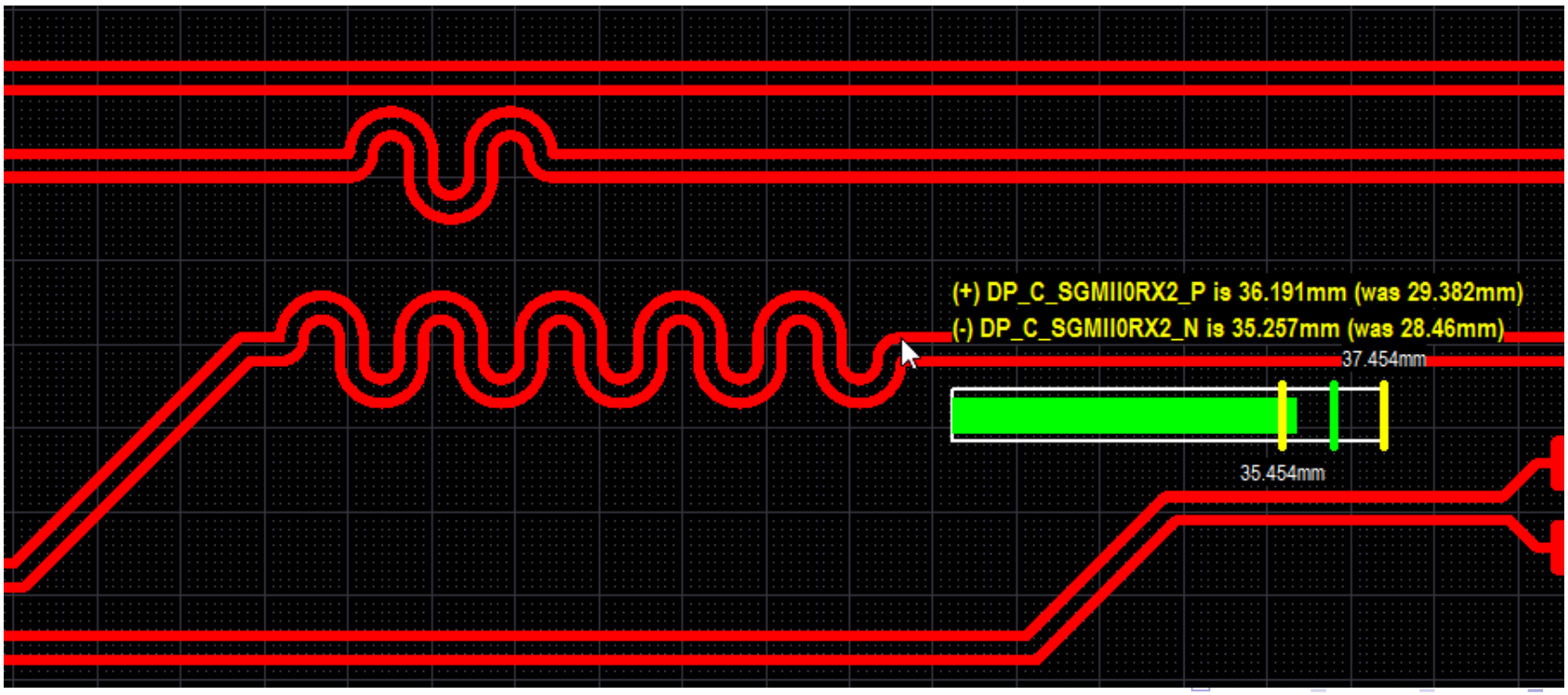
Test Coupons



Differential Pairs

Used on most high speed signals. Why?

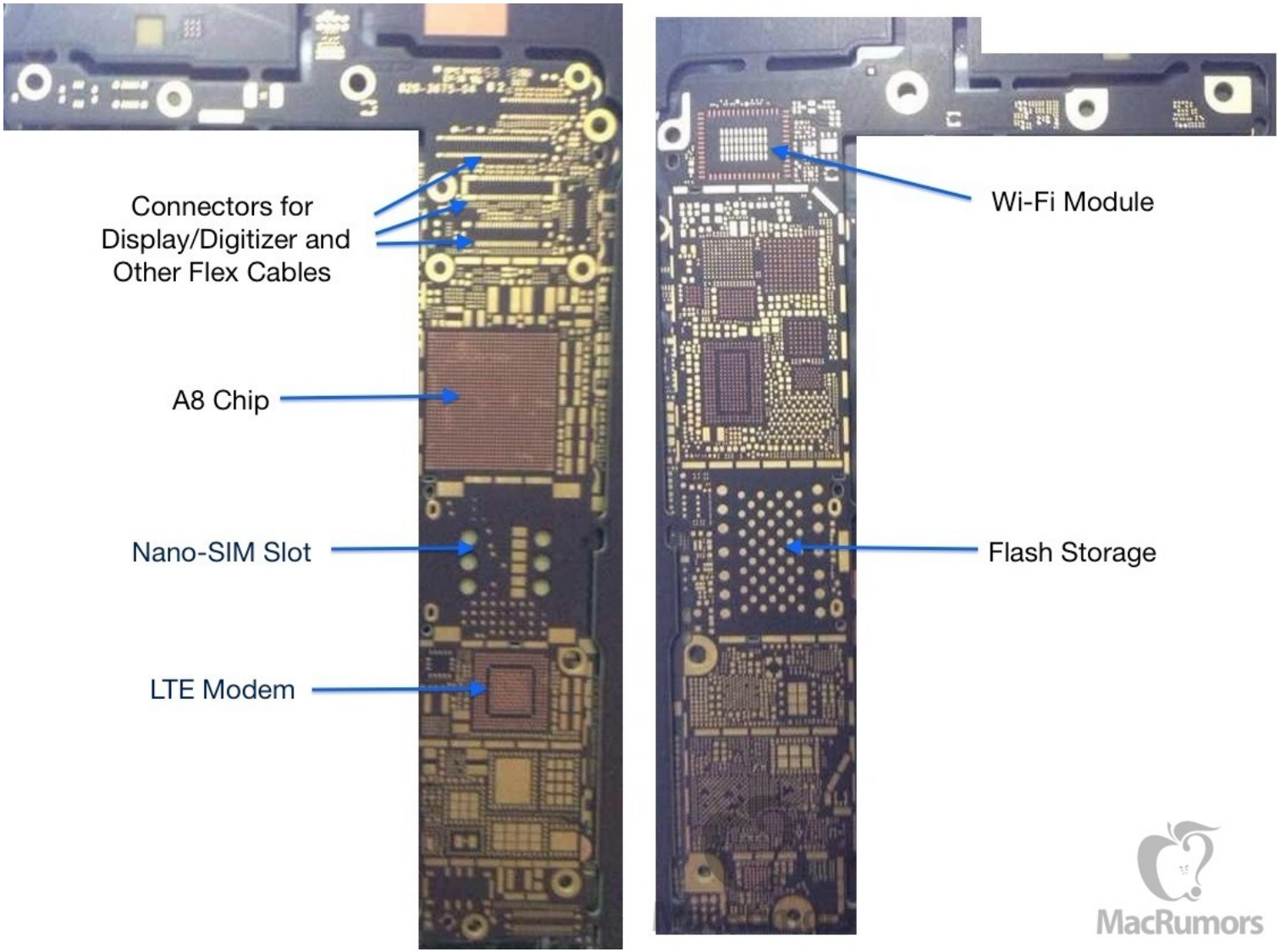
- Minimise crosstalk
- Better functioning in noisy environments
- Reduce electromagnetic interference
- Improvement in SNR
- Isolation from power supplies



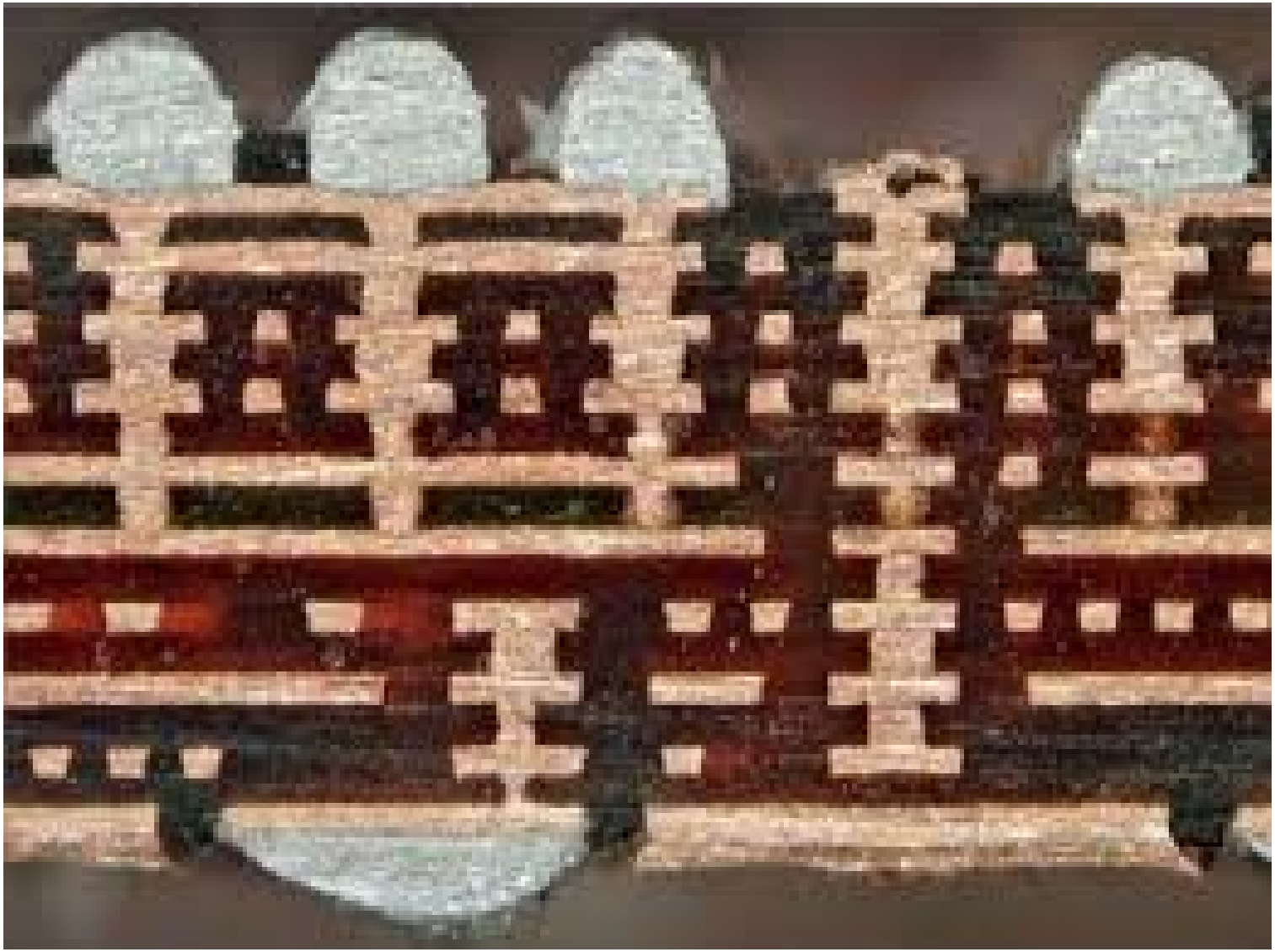
Cutting edge processes

- Blind and buried vias
- Stacked microvias
- Filled and capped vias
- High density interconnect (HDI)
- 2 thou (0.05 mm) trace and space
- Greater than 10:1 aspect ratio vias (6:1 typical)
- Up to 20oz copper (700um, compared to typical 35um)
- Flex and rigid flex
- Distributed element filters
- Embedded passives

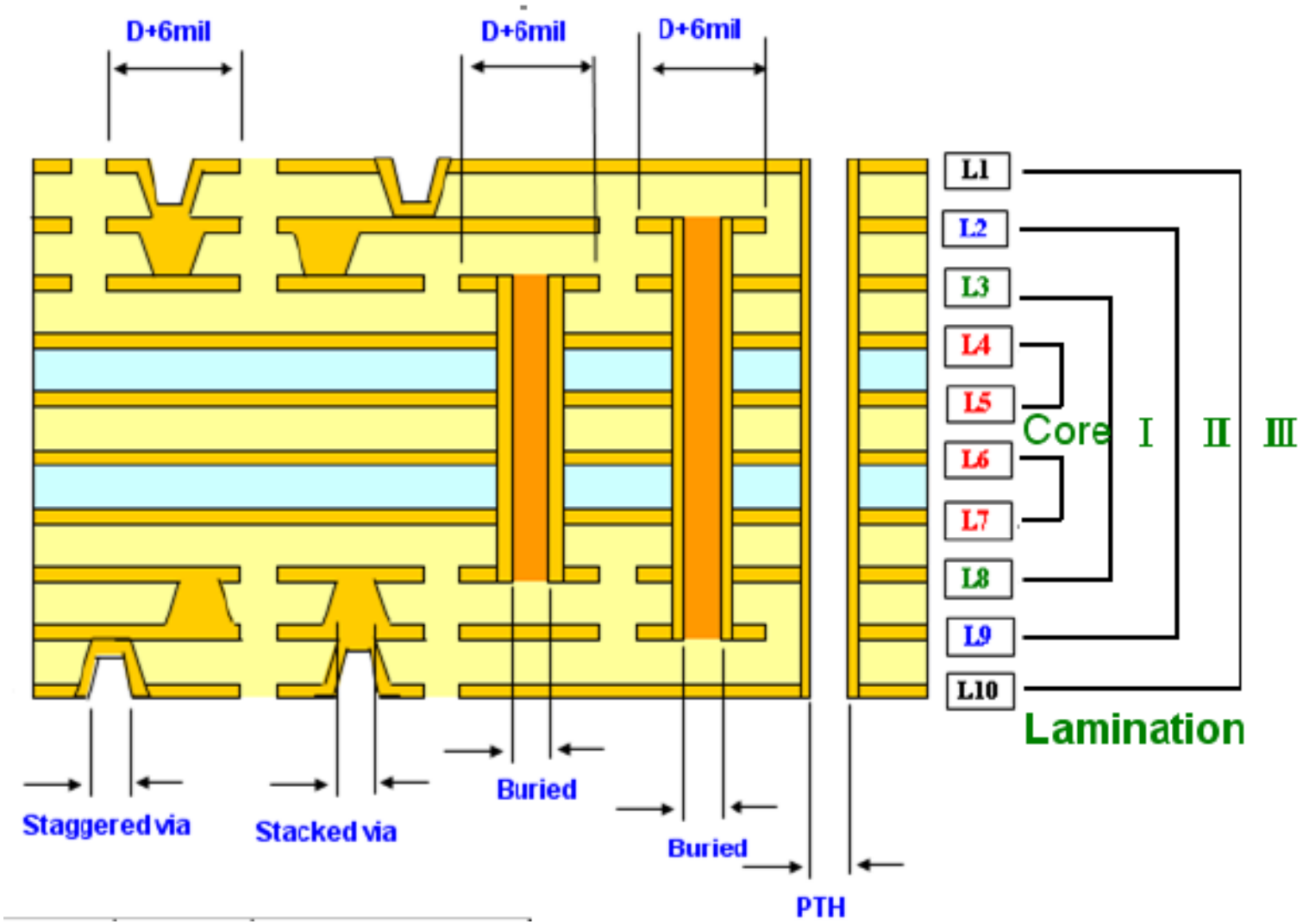
iPhone


Connectors for
Display/Digitizer and
Other Flex Cables
A8 Chip
Nano-SIM Slot
LTE Modem
Wi-Fi Module
Flash Storage
MacRumors

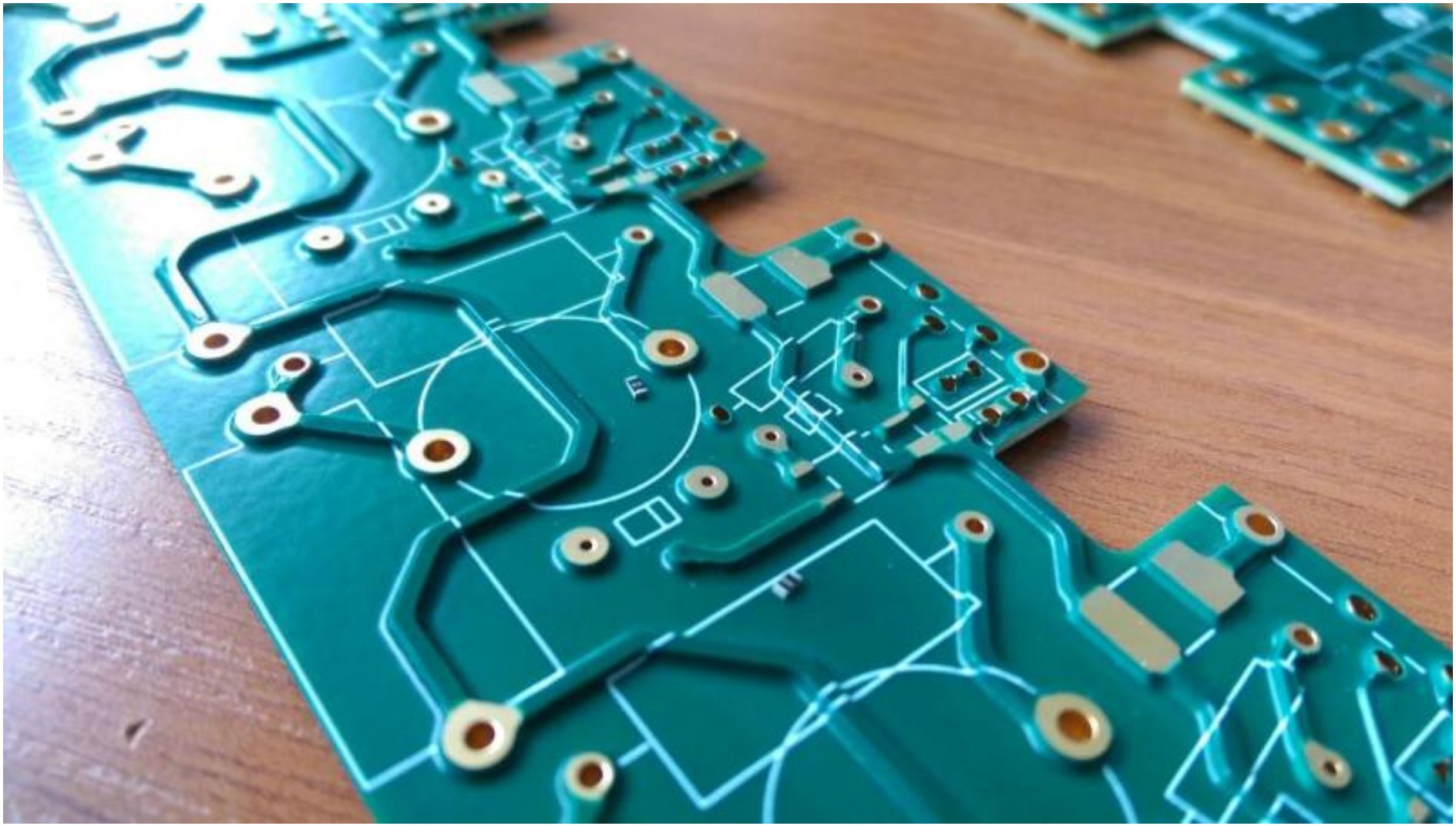
iPhone



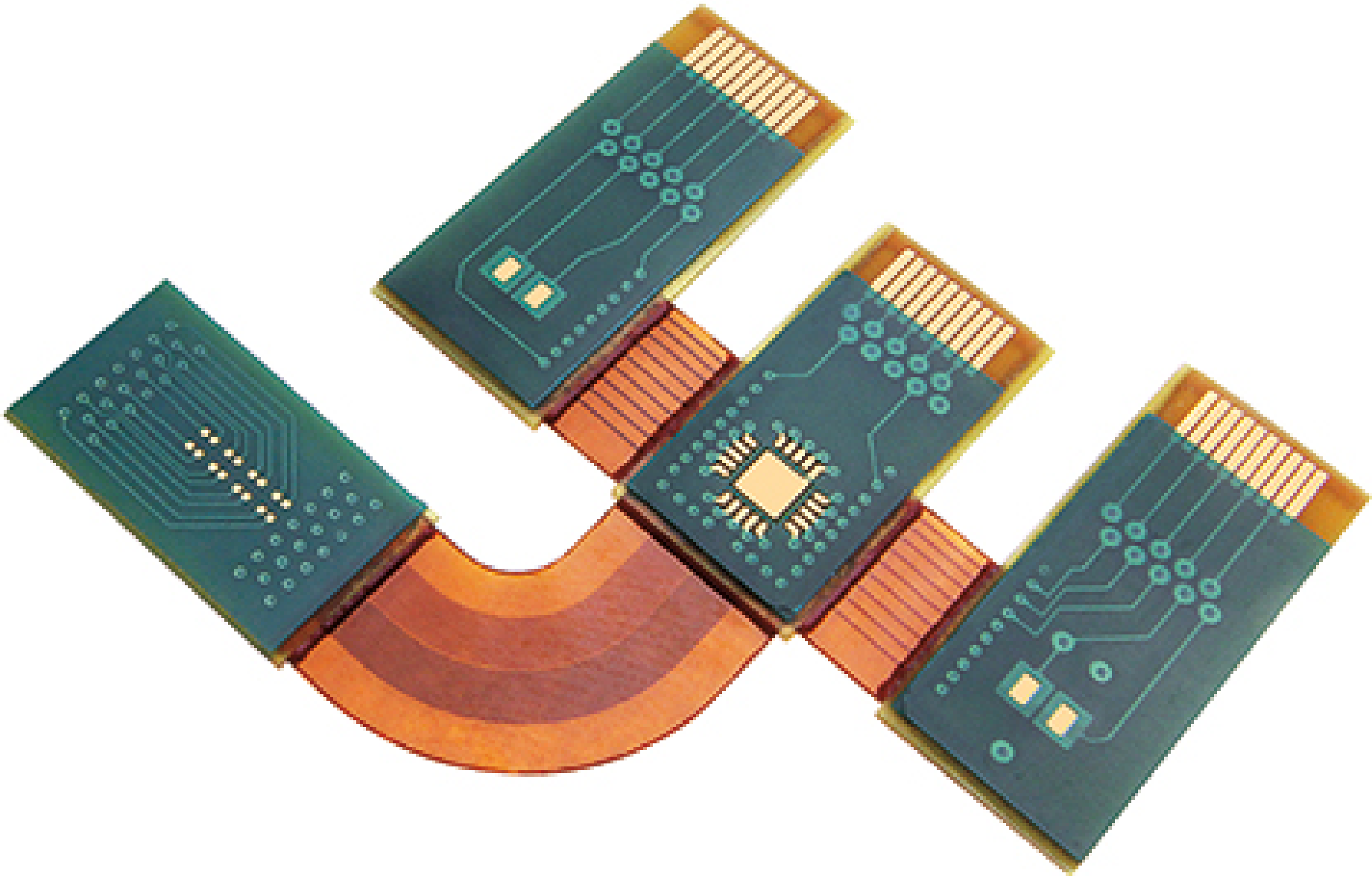
HDI


D+6mil
D+6mil
D+6mil
L1
L2
L3
L4
L5
L6
L7
L8
L9
L10
Core
I
II
III
Lamination
Staggered via
Stacked via
Buried
Buried
PTH

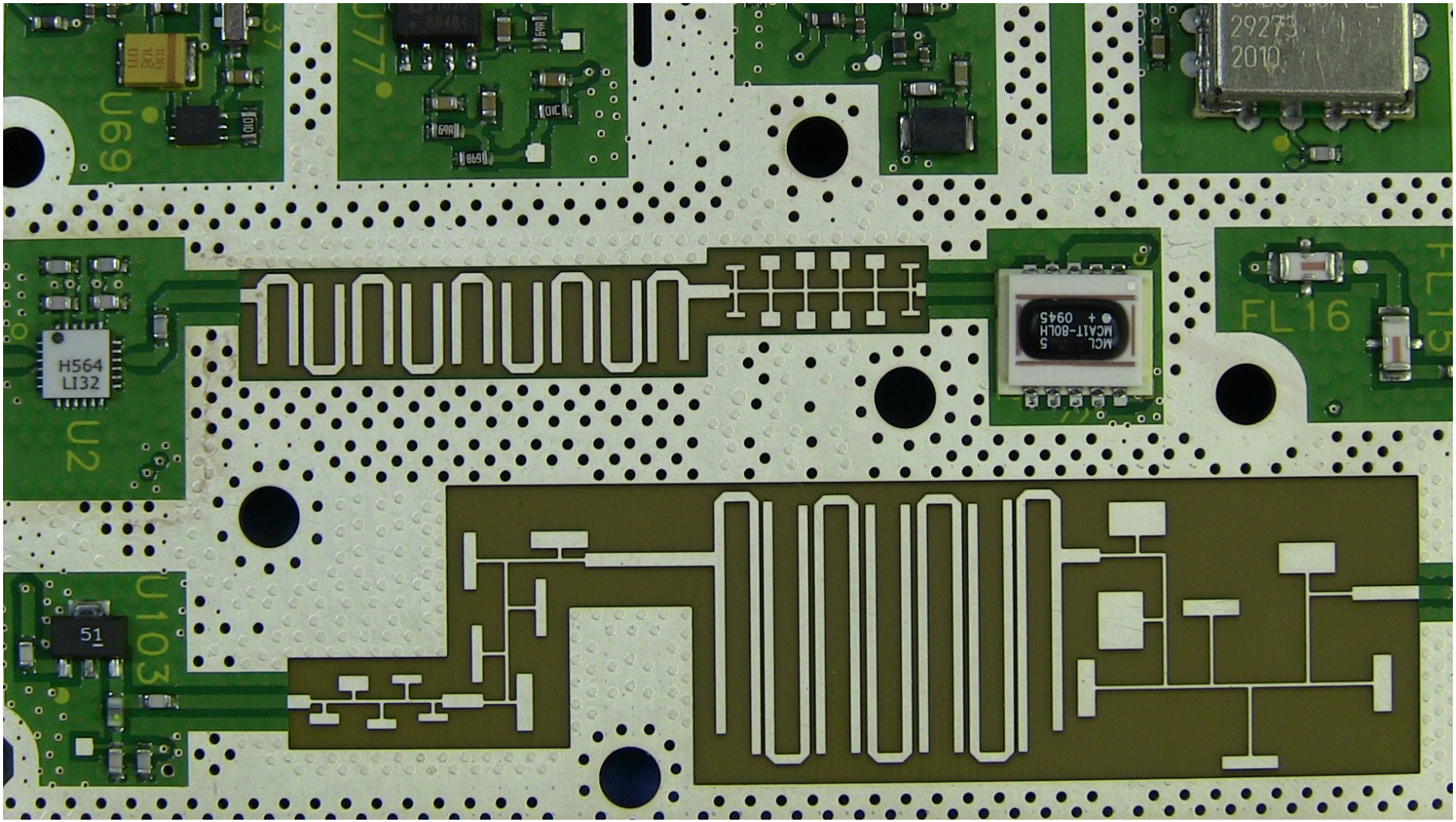
High copper thickness



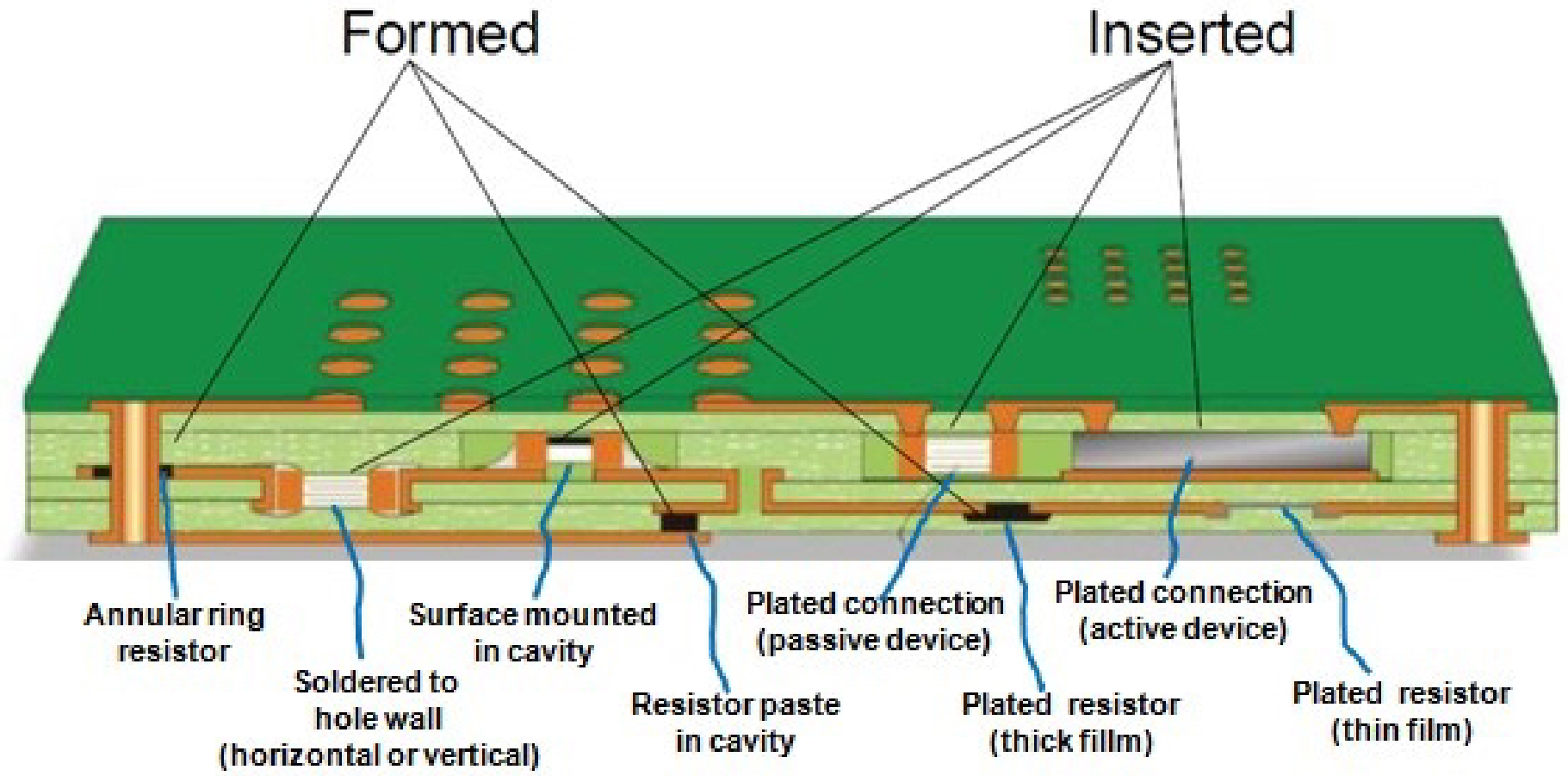
Rigid Flex



Distributed Element Filters



Embedded Passives



Panellisation

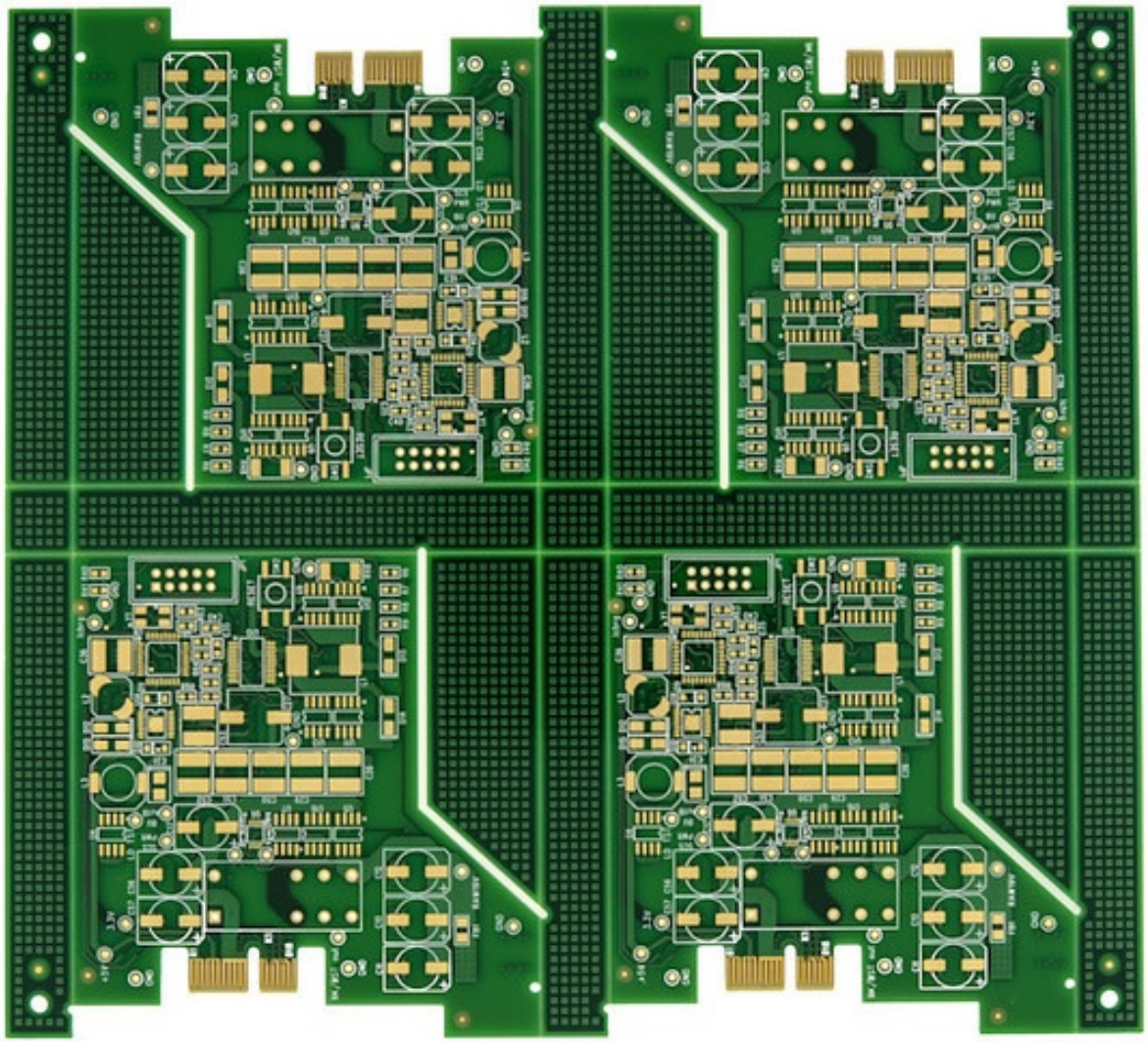
Why do we panellise boards?

- To aide assembly - odd shaped or small boards cannot make it through a SMT line
- To aide programming and test - boards can be gang programmed / tested
- To get more boards for your money - either through sharing tooling charge between multiple people, or getting more boards under a dimension price break

How to break boards out?

- V-Scoring
- Tab and route
- Combination of both

Panellisation



Random KiCad Things

- Library Management
- Panellisation using kicad-util
- Interactive HTML BOM

Layout Tips

- Layout is 80% placement, 20% routing
- Don't push design rules because you can - bigger is better
- Place mounting holes and components first
- Use polygons where practicable
- Keep traces as short at possible
- Route key signals first
- Snake tracks - point to point is not more efficient
- Place decoupling caps close to power pins, smaller value closer
- Series termination resistors close to driver
- Tracks run into center of pads
- No acute angles on traces
- Multiple vias to stitch higher current nets
- Ensure ground / power plane is not cut up, stitch multiple planes together with vias

Layout Review

- 2 layer Keyboard (mechanical constraints)
- 4 layer VNA (RF)
- 4 layer FPGA (somewhat dense)

The End

FPGA Workshop Thoughts?

Say Hello!
BSidesCbr Slack: josh
Twitter: @_joshajohnson
Email: josh@joshajohnson.com

Project Files: `github.com/joshajohnson/CBRhardware`