

# Almost No One Uses The Right PC Board Stack-Up



---

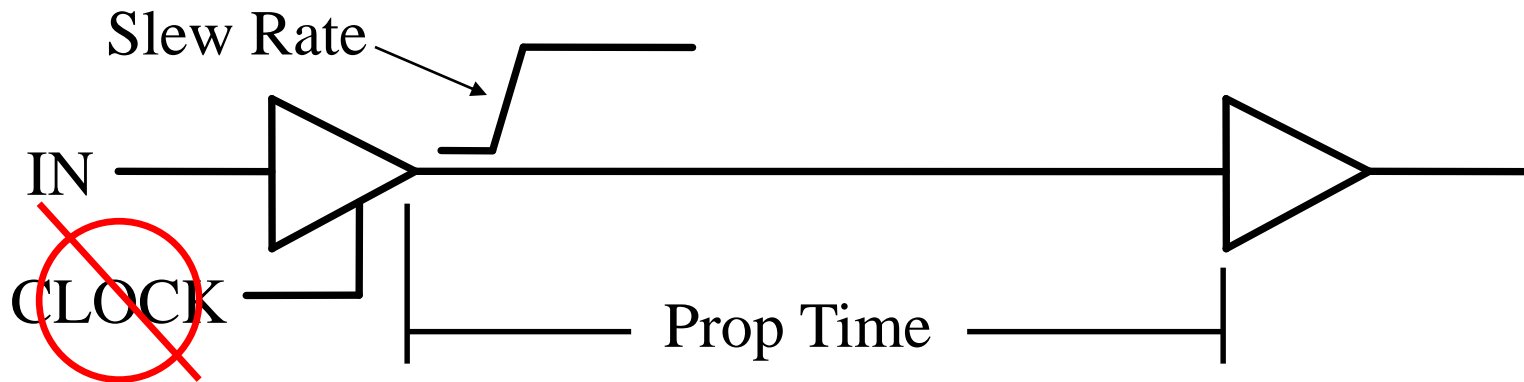
**RHARTLEY**  
ENTERPRISES  
CONTROL OF NOISE, EMI & SI



[rhartley@columbus.rr.com](mailto:rhartley@columbus.rr.com)

# PC Board Stack-Up – Circuit Behavior

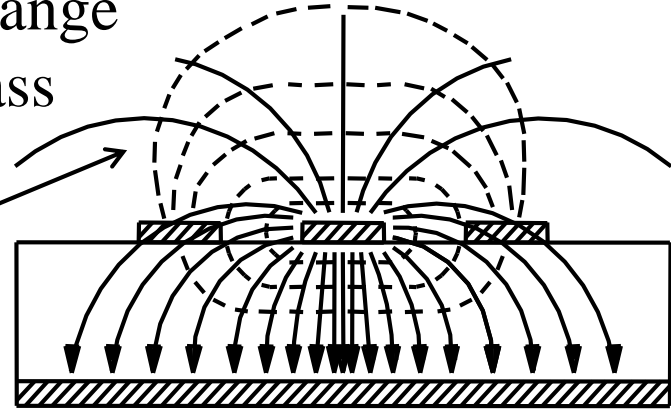
- All signal integrity issues and interference are caused by Fast Rise and Fall IC outputs-



- All Cross Talk, Reflections, Ground Bounce, EMI, etc happen during signal Rise/Fall time.
- Problems are NOT related to Clock Frequency. Clock sets circuit timing margins.

# PC Board Stack-Up – Circuit Behavior

- Inductance is an Impedance to Change in Current flow, caused by the Mass (Inertia) of the Magnetic Field.
- Keep Field volume low, Energy in Field is mostly contained and Inductance will be low.
- Tight coupling between forward and return path are secret to lowering Inductance and to Low Magnetic Field Volume.
- To Keep Capacitance High.....Same thing, tight spacing.
- Doing this minimizes spread of both fields and creates low impedance paths with Low Inductive losses and low spreading of the Fields.



# PC Board Stack-Up – Circuit Behavior

- What is Energy? ... the property of matter that manifests itself as the capacity to perform work (Motion or Interaction of Molecules).
- Energy exists in many forms –
  - Mechanical
  - Sound
  - Light
  - Electrical
  - Heat
  - Etc.



# PC Board Stack-Up – Circuit Behavior

Where is the Energy in a circuit?

- In the Voltage?
- In the Current?
- Neither....  
.... Energy is mostly in the Fields!!!
- Better known as the Electric (E) Field  
and the Magnetic (H or M) Field.



# PC Board Stack-Up – Circuit Behavior

Where are the Fields located in a circuit?

- In the traces?
- In the planes?
- Neither ..... in the space between the traces and the Planes..... in the Dielectric!
- That's Right... the energy in a circuit travels in the plastic and fiberglass material of the PC board, NOT in the Copper!!!



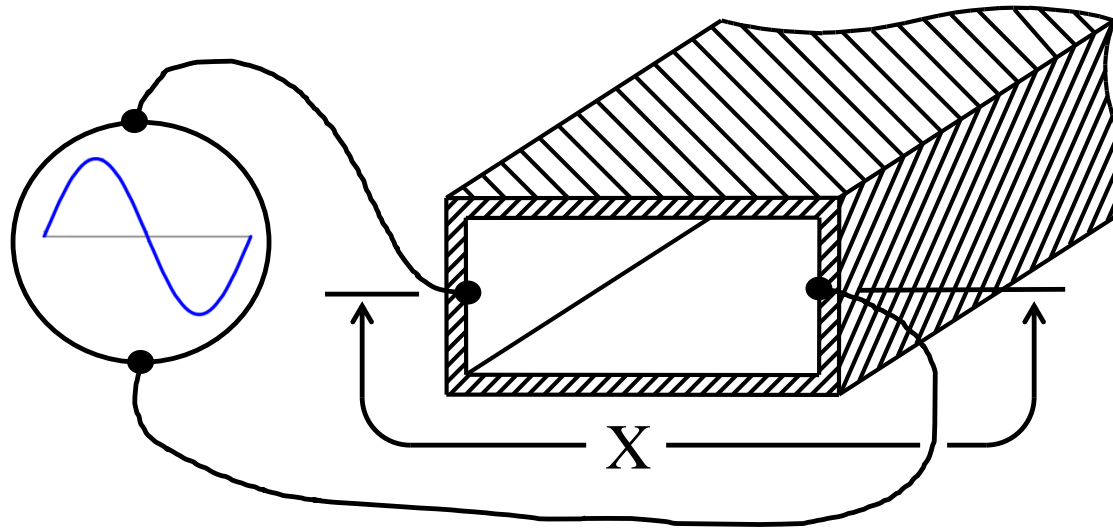
# PC Board Stack-Up – Circuit Behavior

- The energy (E & H/M Fields) in a transmission event is called a Wave, an Electro-Magnetic Wave.
- The traces or the trace and plane that make up the Transmission Line steer the energy from point A to point B.
- These copper elements act as a Wave Guide!



# PC Board Stack-Up – Circuit Behavior

- What is this? ..... A Wave Guide!

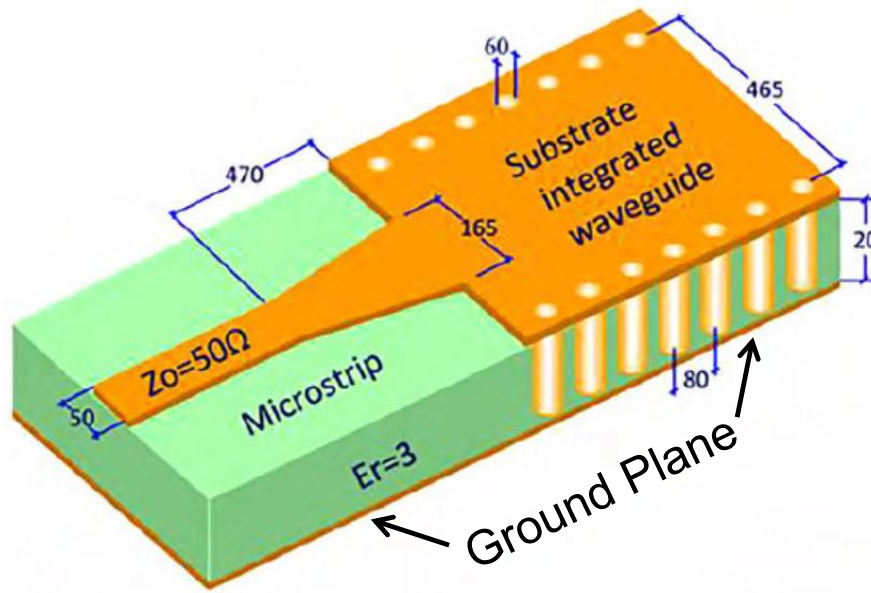


- Apply a Wave whose  $\frac{1}{2}$  wavelength is equal to distance  $X$ , we have perfect balance and a true Wave Guide.



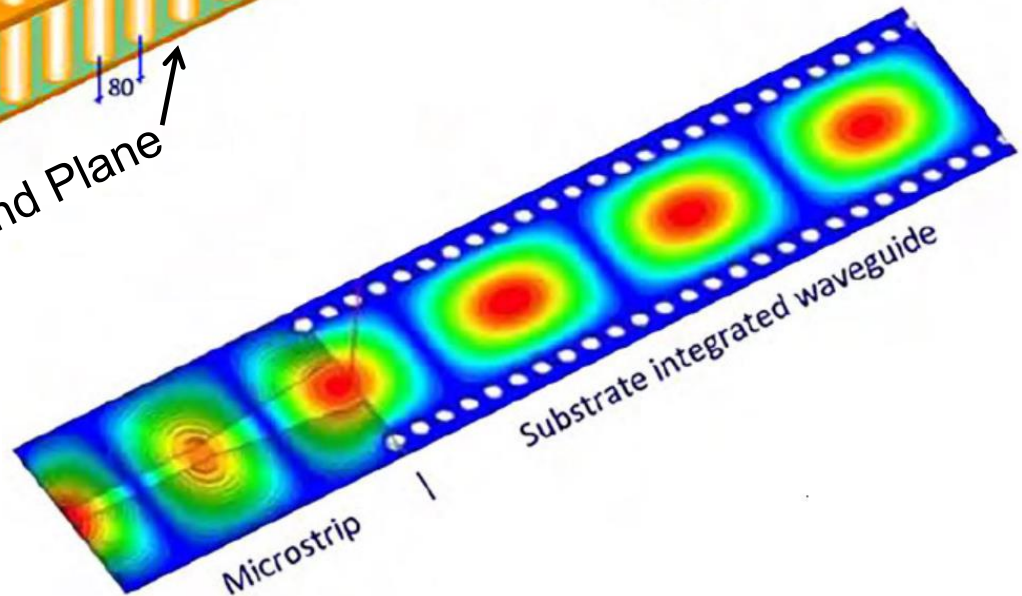
# PC Board Stack-Up – Circuit Behavior

- Microstrip Fed Substrate Integrated Wave Guide-



Source: iCD Pty Ltd (Olney)<sup>1</sup>  
A Review on Substrate Integrated Waveguide and its Microstrip Interconnect (Kumar, Jadhav, Ranade)<sup>2</sup>

Units: Mils  
Copper: 1/4oz  
Not to Scale!



- Both Microstrip and SIW are Wave Guides.



# PC Board Stack-Up – Circuit Behavior

- Whenever we route a trace on some layer of a board, we are routing HALF of a transmission line (Half of a Wave Guide).
- The return side of the line is the return plane or, in some cases, the return trace.
- If we fail to fully understand how to set up the return side of the transmission line, we create field spread and put our circuits in ‘Harms Way’ with regard to interference.

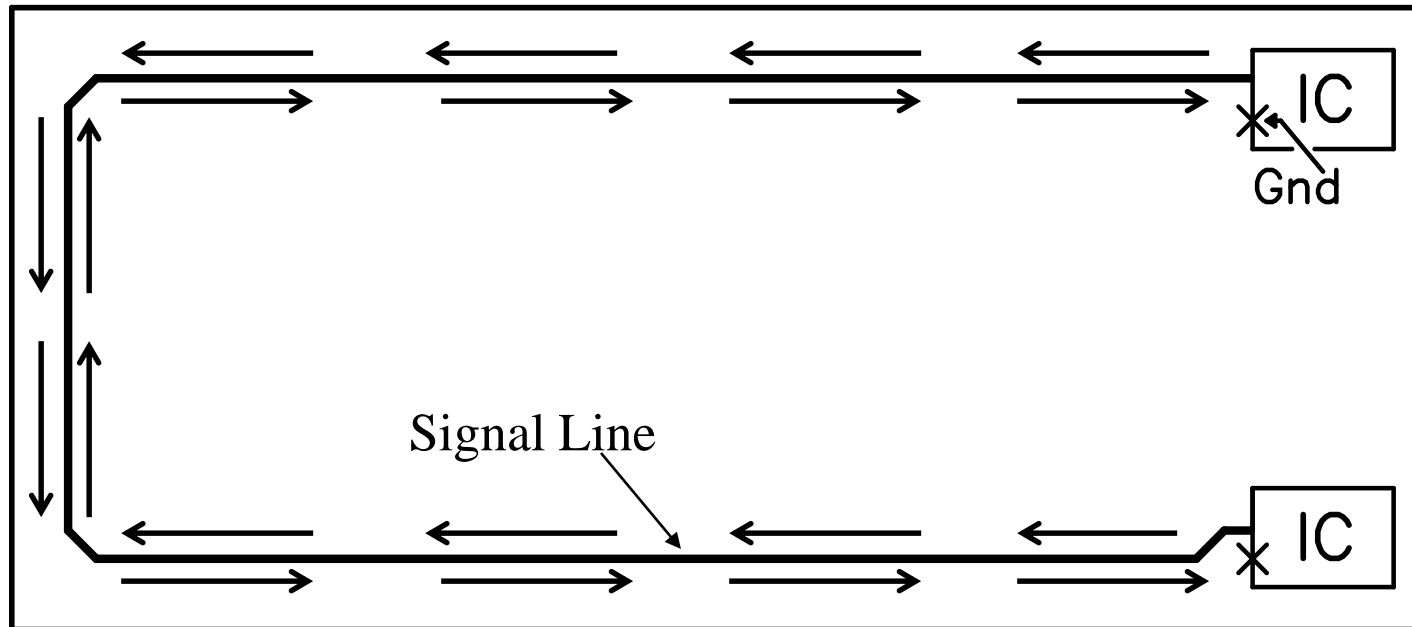


# PC Board Stack-Up – Circuit Behavior

- 2 Layer PC Board -

L1- Routed Signal, routed Power and poured Ground copper.

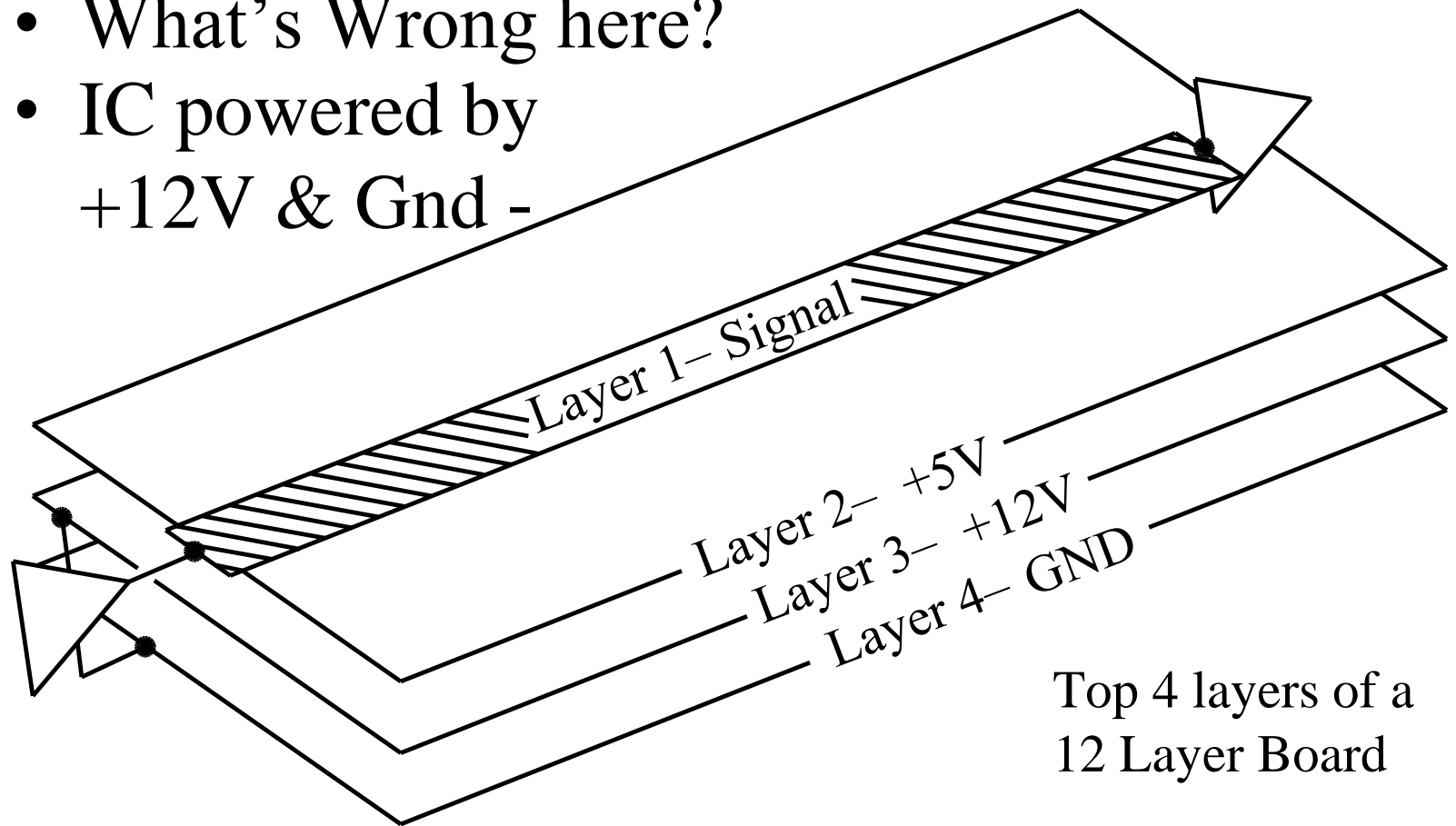
L2- Ground.



Where does signal's return current flow?

# PC Board Stack-Up – Circuit Behavior

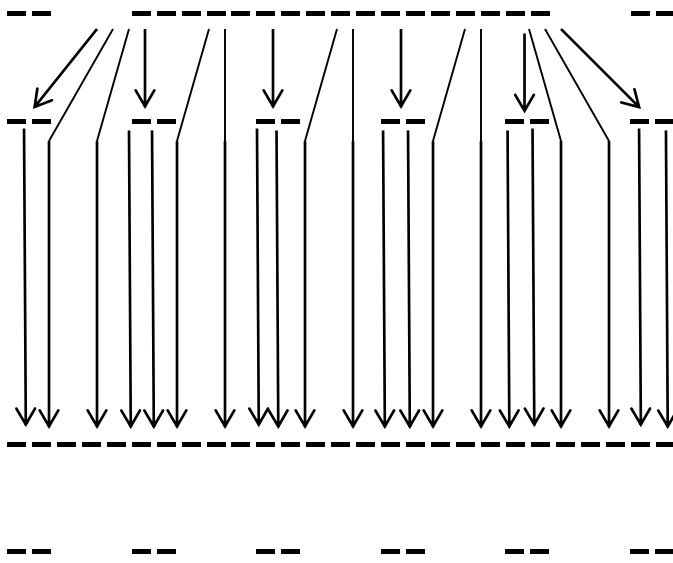
- What's Wrong here?
- IC powered by +12V & Gnd -



Top 4 layers of a  
12 Layer Board

- Where is the Return Current?

# PC Board Stack-Up – Circuit Behavior



Signal – Layer 1

Signal – Layer 2

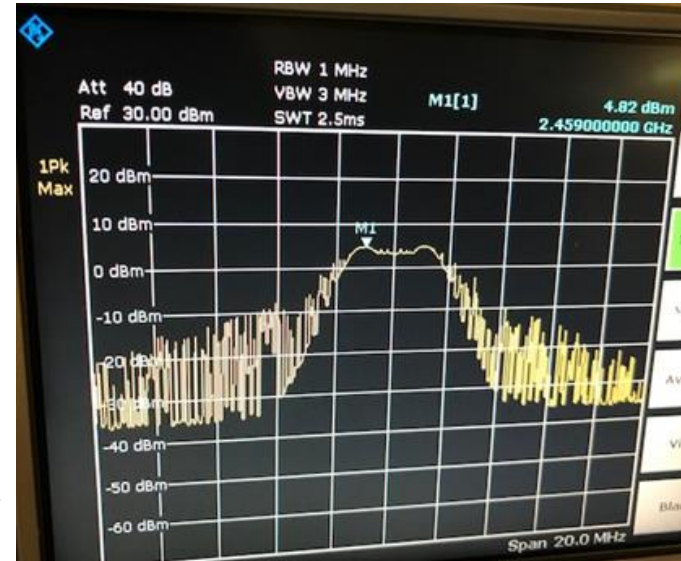
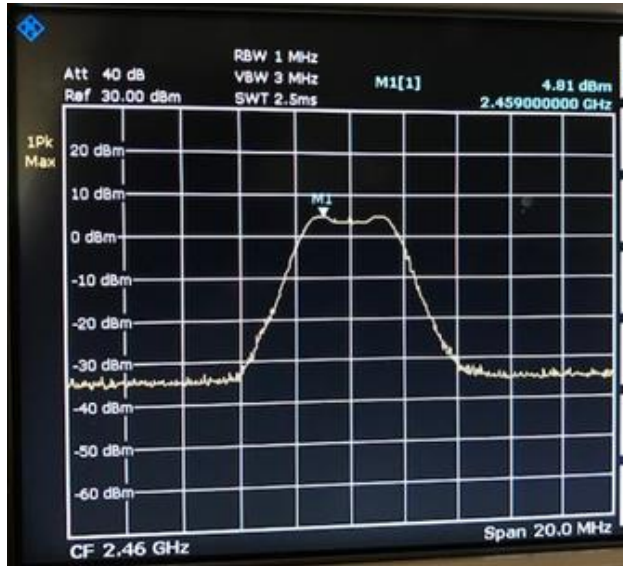
Ground – Layer 3

Signal – Layer 4

- What's Wrong with this Board Stack-up???
- The fields associated with the signals on layer 1 will couple to and thru layer 2 and to the plane on layer 3. ...Serious field Spreading!!!

# PC Board Stack-Up – Circuit Behavior

- Circuit output, when functioning as intended.



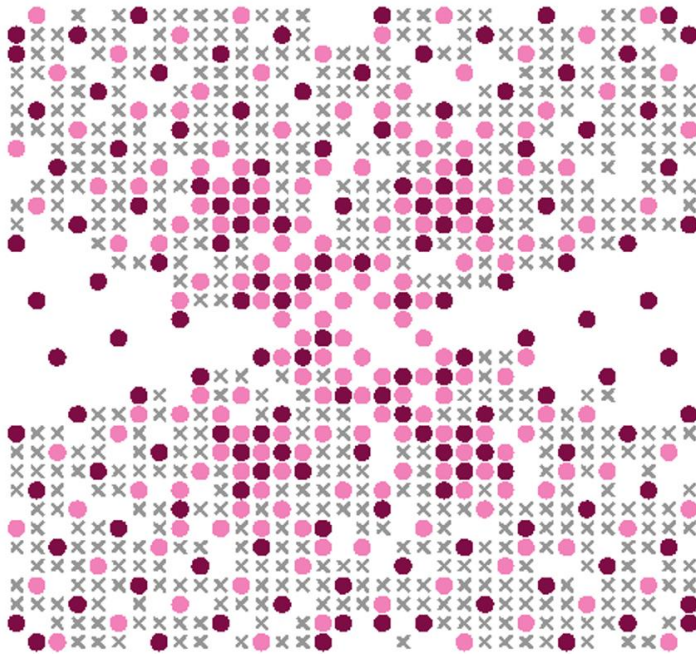
- When second circuit on PCB sampled an accelerometer, output of first circuit looked like this!!!
- PCB used Stack-up on previous page (sig, sig, gnd, sig).
- PCB that eliminated the problem used high quality 6 layer stack-up, described later in presentation.

# PC Board Stack-Up – Circuit Behavior

Though better than many, IC on Right is far less than ideal!!!

F1120 had 5X greater signal interference than FF1148 -

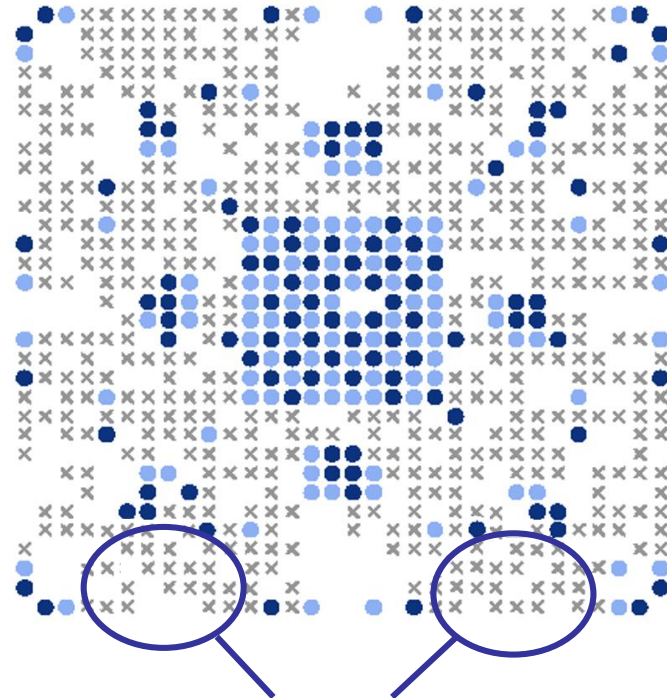
**Xilinx Vertex-4 FF1148**



Returns Spread Evenly

(Source: BGA Crosstalk - Dr. Howard Johnson)

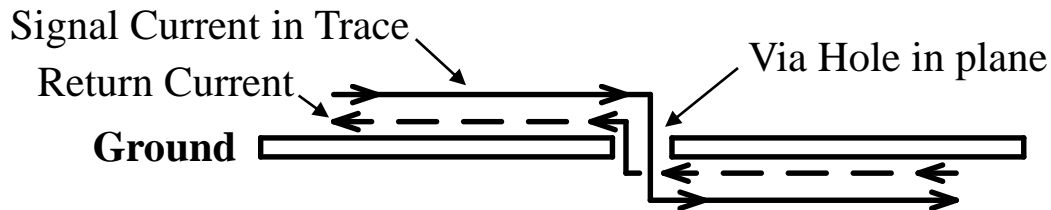
**Altera Stratix-2 F1120**



Many regions devoid  
of Returns

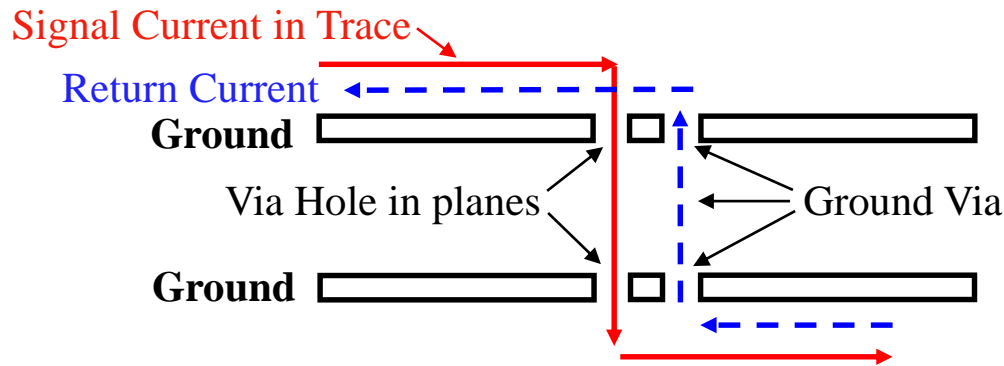
# PC Board Stack-Up – Circuit Behavior

- When moving energy between layers, route on either side of the same plane, as much as possible!!!



Picture:  
Susy Webb

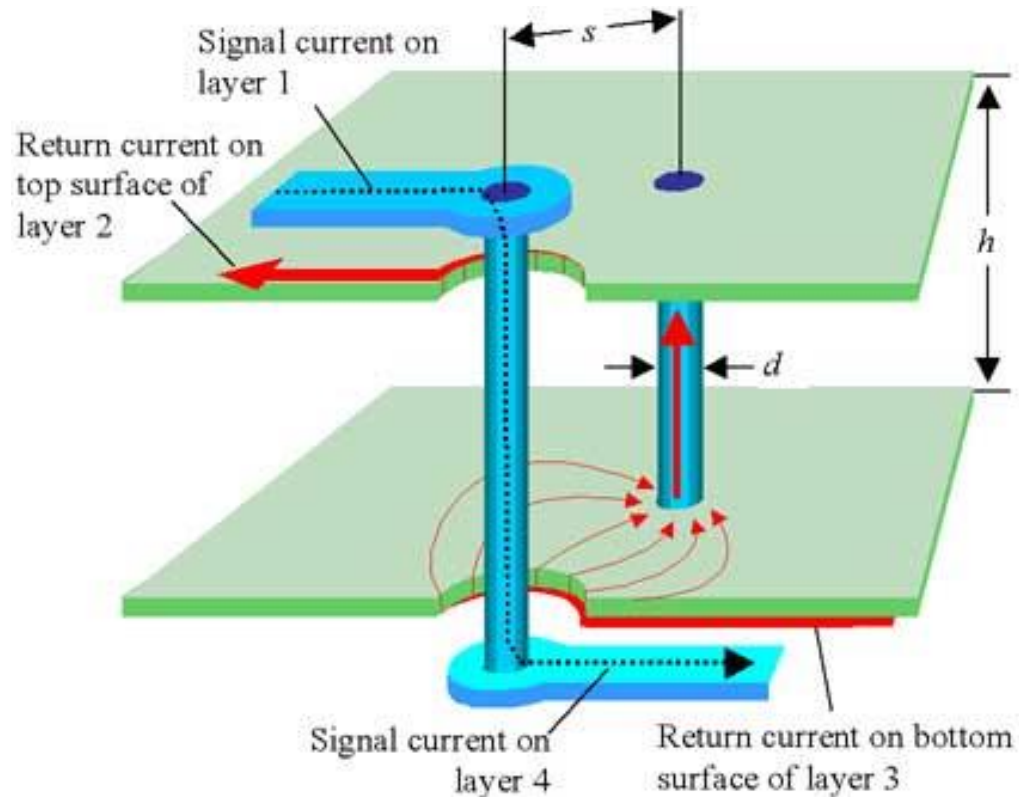
- When moving energy between 2 Ground planes, use a transfer via VERY near the signal via.





# PC Board Stack-Up – Circuit Behavior

With ‘Ground Vias’ Bridging Planes, Loop Area between Forward and Return Path is contained and Field Spread is Minimized!!!

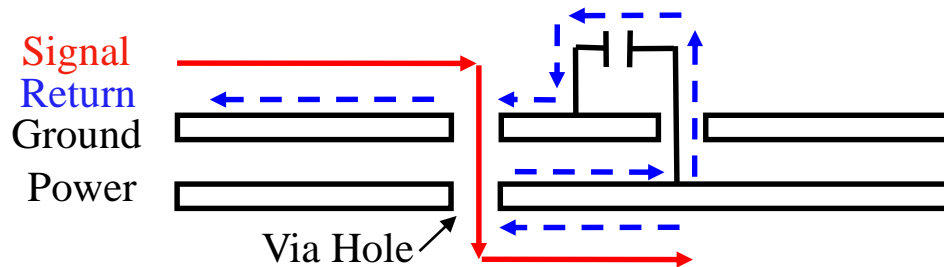


(Source: Dr. Howard Johnson)



# PC Board Stack-Up – Circuit Behavior

- When moving a signal between a Power plane reference and a Ground plane reference –

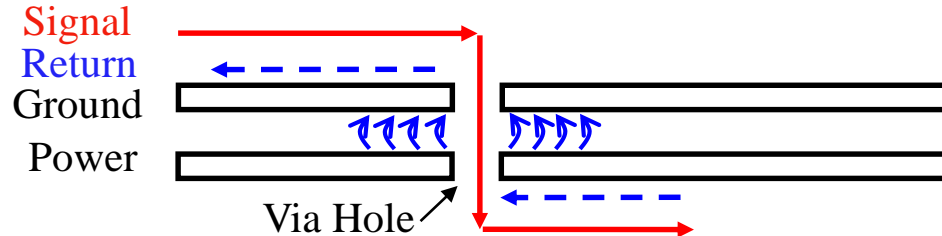


Picture:  
Susy Webb

- Energy below 200 to 300 MHz will travel through the local decoupling capacitors.
- At very high frequencies, Cap is basically an Inductor.
- Beyond 200 to 300 MHz we need to find another path to move energy between dielectric spaces!

# PC Board Stack-Up – Circuit Behavior

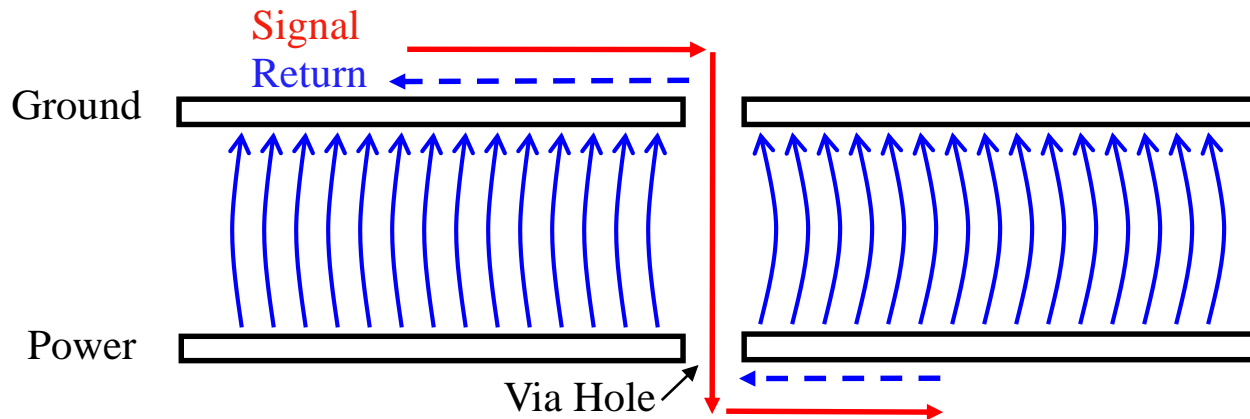
- Where does the energy above 200-300 MHz travel, during a Power-to-Ground layer change?
- Higher frequency ‘returns’ form a displacement current through the natural capacitance formed by the Power and Ground planes.



- This is acceptable when Power-to-Ground plane separation is reasonably small- ie: 0.2mm/.008” or less.... Preferably Less!!!

# PC Board Stack-Up – Circuit Behavior

- When routing signals from Power reference to Ground reference, with widely spaced planes (i.e.- 62 mil thick, 4 Layer Board) -



- To form sufficient displacement current, Fields must spread over a fairly wide area, coupling energy into every via and trace in the area.

# PC Board Stack-Up – Circuit Behavior

-- -- -- -- -- --      Signal – Layer 1

-----  
-----      Power – Layer 2

-----  
-----      Ground – Layer 3

-- -- -- -- -- --      Signal – Layer 4

- What's Wrong with this PCB? Many Things!
- As discussed, Problems when changing layers.
- Thick dielectric creates Power Delivery issues.



# PC Board Stack-Up – Circuit Behavior

- What is solution to the 62 mil thick 4 layer problems?

(A) ----Ground-----  
----Sig/Pwr-----

(B) ---Sig/Poured Pwr---  
-----Ground-----

(Core)

(Core)

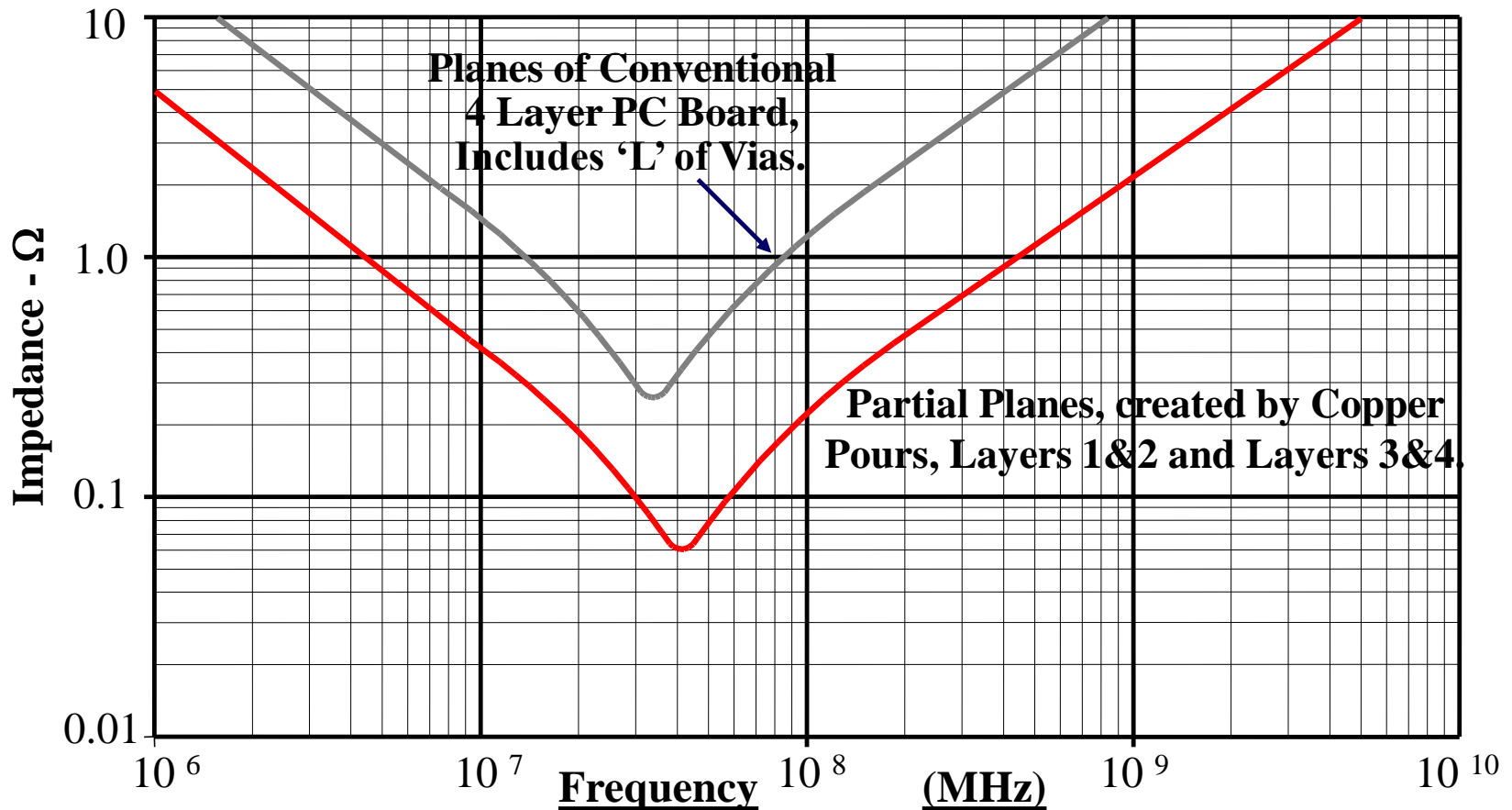
----Sig/Pwr-----  
----Ground-----

-----Ground-----  
---Sig/Poured Pwr---

- Both have design challenges, but vast improvements



# PC Board Stack-Up – Circuit Behavior



**Power Distributed on layers 1&2 and 3&4, creates FAR Lower Impedance than normal 4 layer PCB.**



# PC Board Stack-Up – Circuit Behavior

## 6 Layer PCB to Avoid

-----Signal-----  
-----Power-----  
          (Core)  
-----Signal-----  
-----Signal-----  
          (Core)  
-----Ground-----  
-----Signal-----

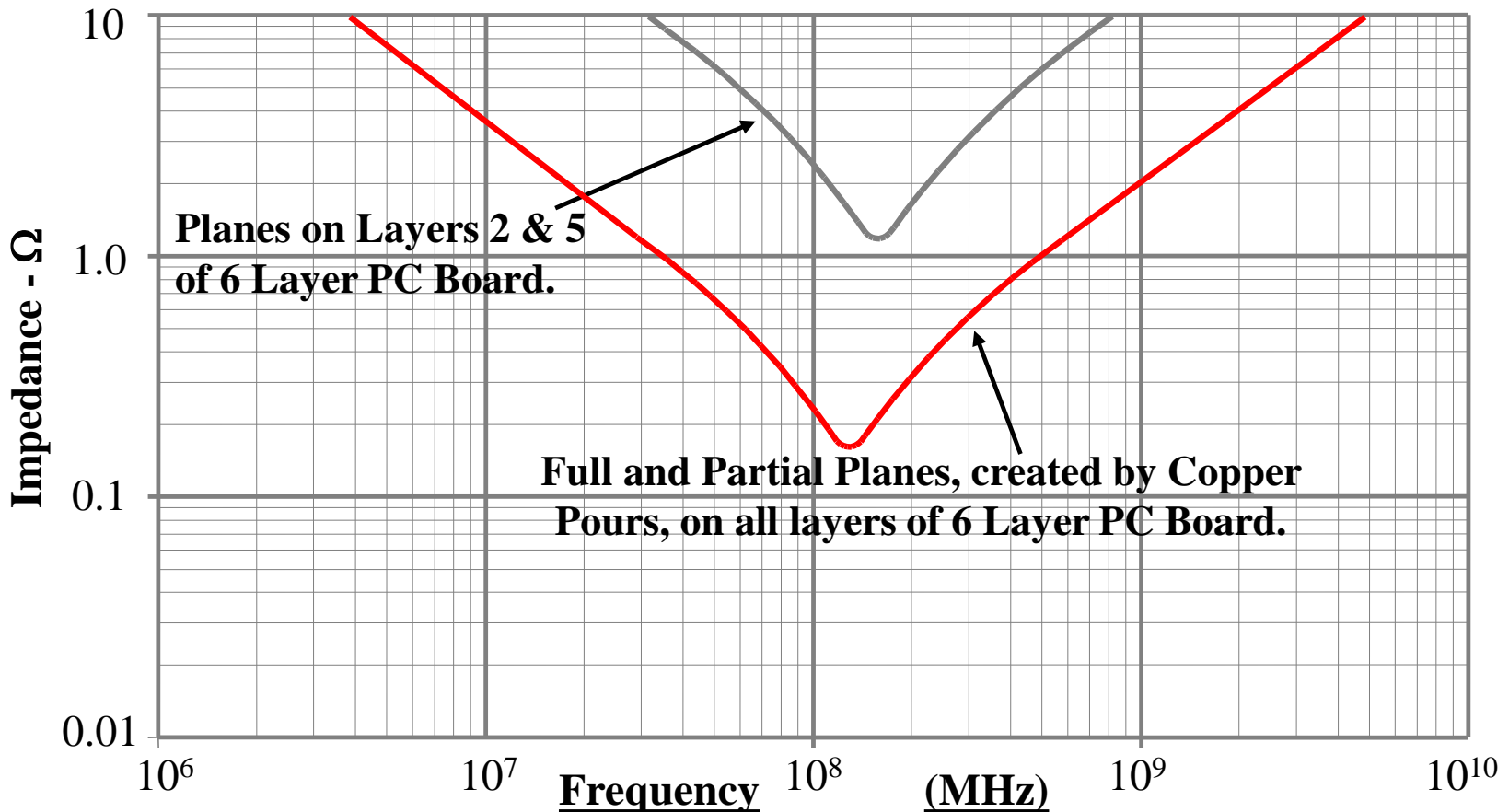
## .062” 6 Layer Solution

-----Sig/Gnd-----  
-----Power-----  
          (Core)  
-----Sig/Gnd-----  
-----Sig/Pwr-----  
          (Core)  
-----Ground-----  
-----Sig/Pwr-----





# PC Board Stack-Up – Circuit Behavior



**Adding Poured Copper on Signal Layers greatly lowers power bus impedance on 6 layer PCB.**



# PC Board Stack-Up – Circuit Behavior

## Even Better 6 Layer

----Sig/Pwr----

----Ground-----

(Core)

----Sig/Pwr----

----Ground-----

(Core)

----Sig/Pwr----

----Ground-----

## And another 6 Layer

----Sig/Pwr----

----Ground-----

(Core)

----Sig/Pwr----

----Sig/Gnd-----

(Core)

----Ground-----

----Sig/Pwr----



# PC Board Stack-Up – Circuit Behavior

8 Layer Stack-up recommended by a MAJOR IC company, for use with a High Speed CPU

----Signal-----

---Ground-----  
(Core)

----Signal-----

---Power 1-----  
(Core)

---Power 2-----

----Signal-----  
(Core)

---Ground-----

----Signal-----

With enough capacitance on Die and Substrate, Signal Integrity could be OK.....

**BUT.....**

Clearly they completed **NO** EMI testing!!!

# PC Board Stack-Up

# Questions???

