Effective PCB Design:

Techniques to Improve Performance

Daniel Beeker
Senior Field Applications Engineer
Freescale Semiconductor, Inc.

Effective PCB Design:

Techniques to improve performance

Smaller device geometries and higher current switching capabilities have thrust us all into the world of RF, HF, UHF, and Microwave *Energy Management*

Rise times on even the lowest tech devices now can exhibit Gigahertz impact.

These changes directly impact product functionality and reliability.

Slide compliments of Ralph Morrison, Consultant

What has changed?

IC technology was described as % shrink from IDR

- Circuit based approach usually was close enough

IC technology now described in nanometers

- Circuit based approach falls completely apart
- EM Field (physics) based approach essential

EMC standards have changed

- Lower frequency compliance requirements
- Higher frequency compliance requirements
- Lower emission levels allowed
- Greater immunity required

The playing field and the equipment have changed!!

This really is a brand new game

What can we do?

The skills required are only taught in a few universities

- Missouri University of Science and Technology, formerly the University of Missouri-Rolla
  - http://www.mst.edu/
- Clemson University
  - http://www.cvel.clemson.edu/emc

Our sagest mentors may not be able to help

Nearly every rule of thumb is wrong!!

To gain the skills needed, you have to actively seek them

Industry Conferences

- PCB East and West
- IEEE EMC Society events

Seminars hosted by your favorite semiconductor supplier!

- Freescale, of course!

What can we do?

There are many “myths” and folklore about the “art” of PCB design

Old “rules of thumb” no longer apply

Time to update our techniques and remove the mystery

Electromagnetic Fields:

The Foundation of Electronics

What is Electricity?

IS IT VOLTS AND AMPERES?

OR

IS IT ELECTRIC AND MAGNETIC FIELDS?

Slide compliments of Ralph Morrison, Consultant

Fields are Basic to ALL Circuit Operation

VOLTS AND AMPERES MAKE THINGS PRACTICAL

- **WE CAN MEASURE VOLTS AND AMPERES NOT E AND H FIELDS**

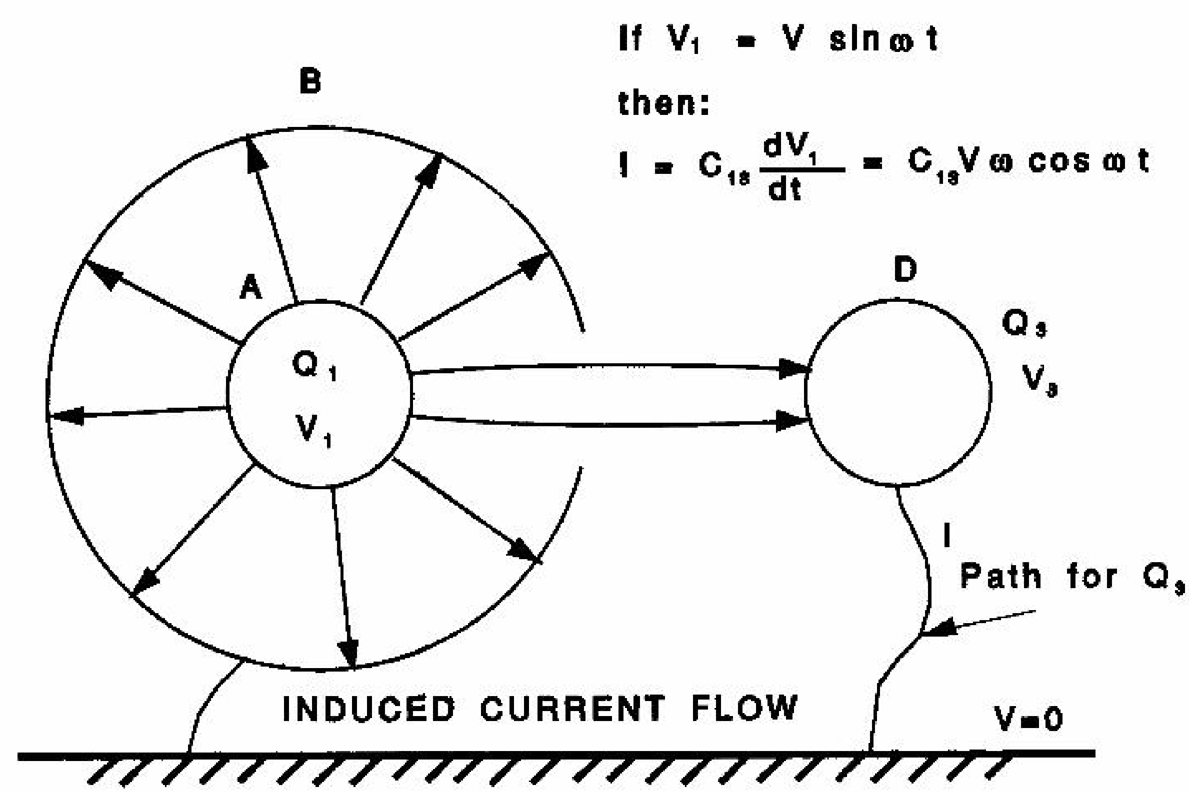
IN HIGH CLOCK RATE (*and* RISE TIME) CIRCUITS, FIELD CONTROL PLAYS A CRITICAL ROLE

THIS MUST BE A CAREFULLY CONSIDERED PART OF ANY DESIGN

Slide compliments of Ralph Morrison, Consultant

Fields go everywhere!

A SHIELD ENCLOSURE WITH A HOLE



Slide compliments of Ralph Morrison, Consultant

ENERGY MANAGEMENT:

Fields store Energy in SPACE!

Energy is NOT stored in or on the conductors

A CAPACITOR IS:
A CONDUCTOR GEOMETRY THAT CONCENTRATES THE STORAGE OF ELECTRIC FIELD ENERGY

AN INDUCTOR IS:
A CONDUCTOR GEOMETRY THAT CONCENTRATES THE STORAGE OF MAGNETIC FIELD ENERGY

IN A CAPACITOR
FIELD ENERGY IS STORED IN THE SPACE BETWEEN THE PLATES

IN AN INDUCTOR
FIELD ENERGY IS STORED IN THE SPACE AROUND WIRES AND IN GAPS

Slide compliments of Ralph Morrison, Consultant

Effective PCB Design:
Techniques to improve performance

Fields behave the same in a component or in space

IN A CAPACITOR:

A CHANGING VOLTAGE MEANS THE E FIELD IS CHANGING AND THAT CURRENT IS FLOWING

IN SPACE:

A CHANGING E FIELD IS A DISPLACEMENT CURRENT THIS CURRENT CREATES A MAGNETIC FIELD

All components require fields to operate

FIELDS CARRY ENERGY - NOT CONDUCTORS

WHAT ARE THE CONDUCTORS FOR?

THEY TELL THE ENERGY WHERE TO GO!

Slide compliments of Ralph Morrison, Consultant

Why does Energy follow conductors?

WHY DOES WATER FLOW IN A STREAM?
SAME REASON

NATURE FOLLOWS THE PATH THAT STORES THE LEAST ENERGY

IT IS EASIER FOR FIELDS TO *FOLLOW* TRACES THAN TO GO OUT ACROSS SPACE

Slide compliments of Ralph Morrison, Consultant

Transmission Lines are Convenient paths for Energy flow

EVERY CONDUCTOR PAIR IS A TRANSMISSION LINE

TRACE-TO-TRACE OR TRACE-TO-CONDUCTING PLANE

THE FIELDS, AND THUS THE ENERGY FLOW, WILL CONCENTRATE BETWEEN TRACES OR BETWEEN A TRACE AND A CONDUCTING PLANE

DRAW THE FIELDS TO LOCATE THE CURRENT

Slide compliments of Ralph Morrison, Consultant

Properties of Transmission Lines

THEY DIRECT ENERGY FLOW
THEY CAN STORE FIELD ENERGY
THEIR POSITION IN A CIRCUIT IS CRITICAL
THEY CROSS COUPLE ENERGY ONLY AT WAVE FRONTS
THEY DELIVER ENERGY AT TERMINATIONS
THEY ARE BI-DIRECTIONAL
THEY CAN TRANSPORT ANY NUMBER OF WAVES AT ONE TIME
THEY CAN RADIATE

Slide compliments of Ralph Morrison, Consultant

We Use Transmission Lines to Transport Energy and to Carry Logic Signals

A TRANSMISSION LINE CAN CARRY ANY NUMBER OF SIGNALS IN EITHER DIRECTION AT THE SAME TIME

BELOW 1 MHz THE GEOMETRY OF THESE LINES IS NOT TOO CRITICAL

WITH TODAY'S CLOCK RATES AND RISE TIMES THE GEOMETRY OF THESE LINES IS KEY TO PERFORMANCE

In a good design:

FIELDS ASSOCIATED WITH DIFFERENT SIGNALS DO NOT SHARE THE SAME PHYSICAL SPACE.

IF THEY DO SHARE THE SAME SPACE, THERE IS CROSSTALK!

Slide compliments of Ralph Morrison, Consultant

In a good design:

ENERGY IS AVAILABLE WHENEVER THERE IS A DEMAND
THE VOLTAGE SOURCE MUST BE REASONABLY CONSTANT
ENERGY MUST BE REPLACED AFTER IT IS USED OR THERE WILL BE LOGIC PROBLEMS
THIS IS CALLED ENERGY MANAGEMENT

Local Sources of Energy:

DECOUPLING CAPACITORS
THERE IS ALSO ENERGY AVAILABLE FROM THE GROUND/POWER PLANE CAPACITANCE

New Problem:

IT TAKES TIME TO MOVE THIS ENERGY FROM STORAGE TO A LOAD

Slide compliments of Ralph Morrison, Consultant

How long does it take??

Wave Velocity

FOR TRACES ON A CIRCUIT BOARD $v = c / € ^{1/2}$
WHERE c I S THE VELOCITY OF LIGHT AND € IS THE RELATIVE DIELECTRIC CONSTANT

v = 150 mm / ns or 6” / ns

All Energy is moved by Wave Action!!

A DROP IN VOLTAGE SENDS A WAVE TO GET MORE ENERGY
WAVES REFLECT AT DISCONTINUITIES
A SOURCE OF VOLTAGE IS A DISCONTINUITY
EACH REFLECTED WAVE CAN CARRY A LIMITED AMOUNT OF ENERGY

Slide compliments of Ralph Morrison, Consultant

What does this mean in my circuit board?

Initial power level in a 50 ohm line

5 OHM LOAD AND 5 V SOURCE **I = 0.1 AMPERES OR ½ WATT**

Now, how do I get 1 Ampere?

EVEN IF THE LINE IS ONLY 1/16 INCH LONG:

IT TAKES 10 ps FOR A WAVE TO GO 1/16 INCH IN FR4

IT TAKES 20 ps FOR A WAVE TO MAKE ONE ROUND TRIP

IT TAKE 30 ROUND TRIPS ON THAT LINE TO BRING THE CURRENT LEVEL UP TO NEAR 1 AMP

THAT IS 600 ps, ASSUMING ZERO RISE TIME

Slide compliments of Ralph Morrison, Consultant

Typical 1/16 inch connections:

Traces to CAPACITORS

CONNECTIONS to IC DIES

Lead frames and wire bonds

BGA interposers

Traces to VIAs

VIAs to GROUND/POWER PLANES

Slide compliments of Ralph Morrison, Consultant

Capacitors are Short Transmission Lines!

WAVE ACTION IS REQUIRED TO MOVE ENERGY IN AND OUT OF A CAPACITOR

Don't forget the connections to the capacitor!

SELF INDUCTANCE DOES NOT PROPERLY TELL THE STORY OF WHY IT TAKES TIME TO SUPPLY ENERGY

CIRCUIT THEORY DOES NOT CONSIDER TIME DELAYS

Slide compliments of Ralph Morrison, Consultant

All Energy is moved by Wave Action!!

When a switching element closes, this results in a drop in the voltage on the power supply. The resulting field energy *request* wave travels until this request is filled or it radiates.

The only way to reduce noise in a system is to reduce this distance and provide adequate sources of Electromagnetic Field energy.

Energy source hierarchy:

On-Chip Capacitance
Power Planes if present
Local bypass capacitors
Field energy stored across the PCB structure
Bulk storage capacitors
Finally the power supply

Providing adequate "FIELD ENERGY" in a timely manner is essential to reducing system noise!

Slide comments are compliments of Ralph Morrison, Consultant

Antenna size vs. Frequency

Frequency	¼ wave length
1 Hertz Rise time equivalent, who cares	246,000,000 feet (46,591 miles) Almost 6 times around the earth
10 Hertz Rise time equivalent, still who cares	24,600,000 feet (4,659 miles) Almost from New York to Honolulu
100 Hertz Rise time equivalent, .01 seconds	2,460,000 feet (466 miles) Almost from New York to Detroit
1 KHz Rise time equivalent, 1 millisecond	246,000 feet (46.6 miles) Almost from Orlando to Cocoa Beach
10 KHz Rise time equivalent, 100 microseconds	24,600 feet (4.659 miles) Almost from the J. W. Marriott to Disney's Magic Kingdom
100 KHz Rise time equivalent, 10 microseconds	2,460 feet (0.466 miles) Almost from the J. W. Marriott to the Central Florida Parkway
1 MHz Rise time equivalent, 1 microsecond	246 feet (0.0466 miles) Less than a football field
10 MHz Rise time equivalent, 100 nanoseconds Rise time distance, 100 feet	24.6 feet Across the room
100 MHz (TTL Logic) Rise time equivalent, 10 nanoseconds Rise time distance, 10 feet	2.46 feet Less than a yard
1 GHz (BiCMOS Logic) Rise time equivalent, 1 nanosecond Rise time distance, 1 foot	0.246 feet (2.952 inches) Less than your finger
10 GHz (GaAs Logic) Rise time equivalent, 100 picoseconds Rise time distance, 1.2 inches	0.0246 feet (0.2952 inches) Less than the diameter of a pencil
100 GHz (nanometer geometry HCMOS) Rise time equivalent, 10 picoseconds Rise time distance, 0.12 inches	0.00246 feet (0.0295 inches) Half the thickness of a standard FR4 PCB

Antenna size vs. Frequency

Frequency	¼ wave length
10 MHz HMOS Rise time equivalent, 100 nanoseconds Rise time distance, 100 feet	24.6 feet Across the room
100 MHz (TTL Logic) UDR HCMOS Rise time equivalent, 10 nanoseconds Rise time distance, 10 feet	2.46 feet Less than a yard
1 GHz (BiCMOS Logic) IDR HCMOS Rise time equivalent, 1 nanosecond Rise time distance, 1 foot	0.246 feet (2.952 inches) Less than your finger
10 GHz (GaAs Logic) 65 nM HCMOS Rise time equivalent, 100 picoseconds Rise time distance, 1.2 inches	0.0246 feet (0.2952 inches) Less than the diameter of a pencil
100 GHz 32 nM HCMOS Rise time equivalent, 10 picoseconds Rise time distance, 0.12 inches	0.00246 feet (0.0295 inches) Half the thickness of a standard FR4 PCB

Effective PCB Design:

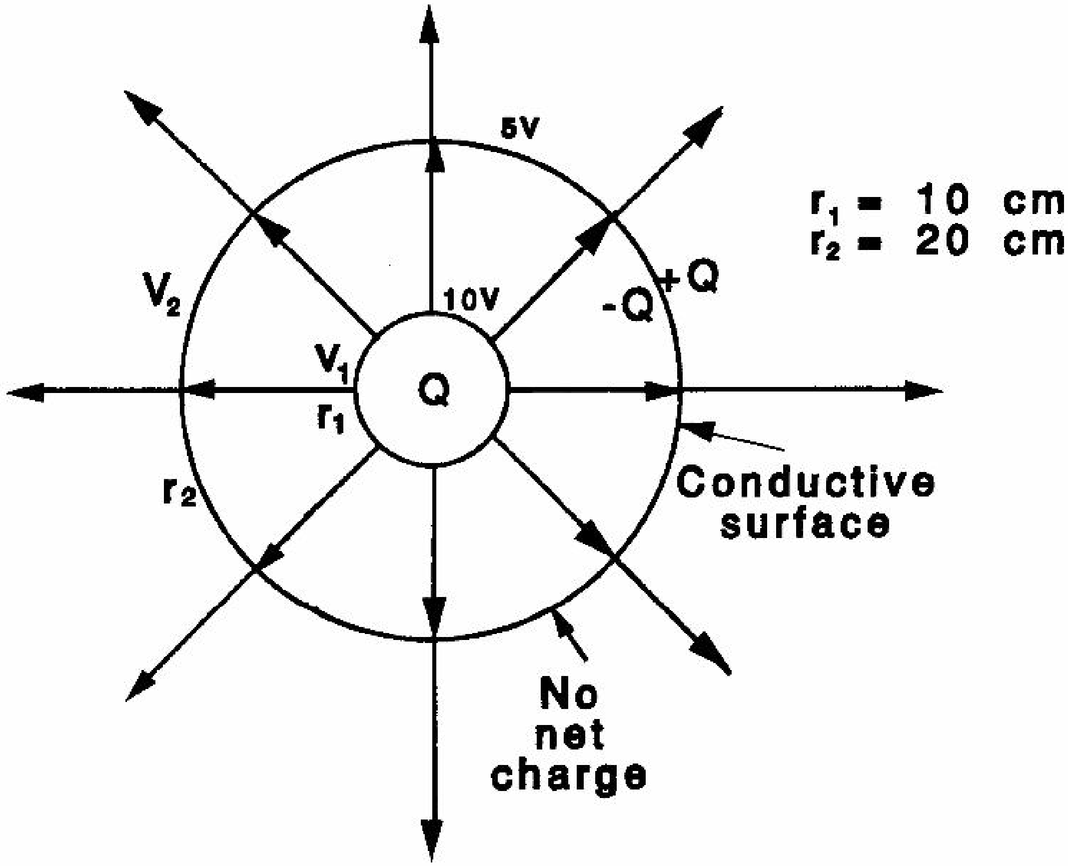
Techniques to improve performance

From the previous table, a few things become apparent:

- We got away with ignoring basic physics because IC switching speeds were *slow* and efficient antennas had to be **HUGE**.
- At a switching speed of 1 nanosecond, it only takes a PCB feature (trace or slot) of 3 inches to be an efficient antenna (1/4 wave length)
- Once you cross that magic boundary of 1 nanosecond, most PCB designs are capable of providing a wonderful source of antennas
- At 10 picosecond speeds, almost every structure on a PCB can be an good antenna

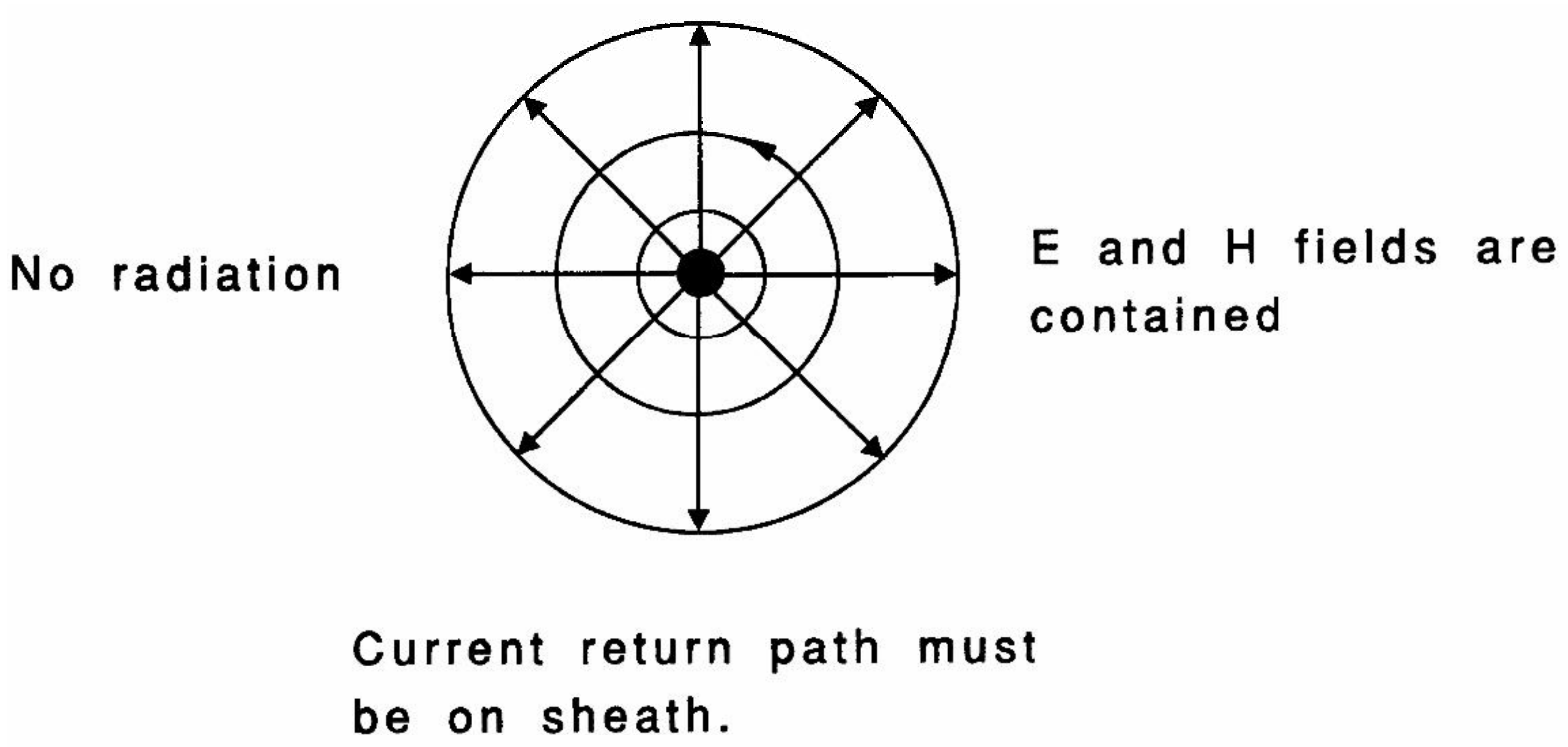
Fields are friendly!

AN EQUIPOTENTIAL SURFACE AROUND A CHARGED SPHERE



Slide compliments of Ralph Morrison, Consultant

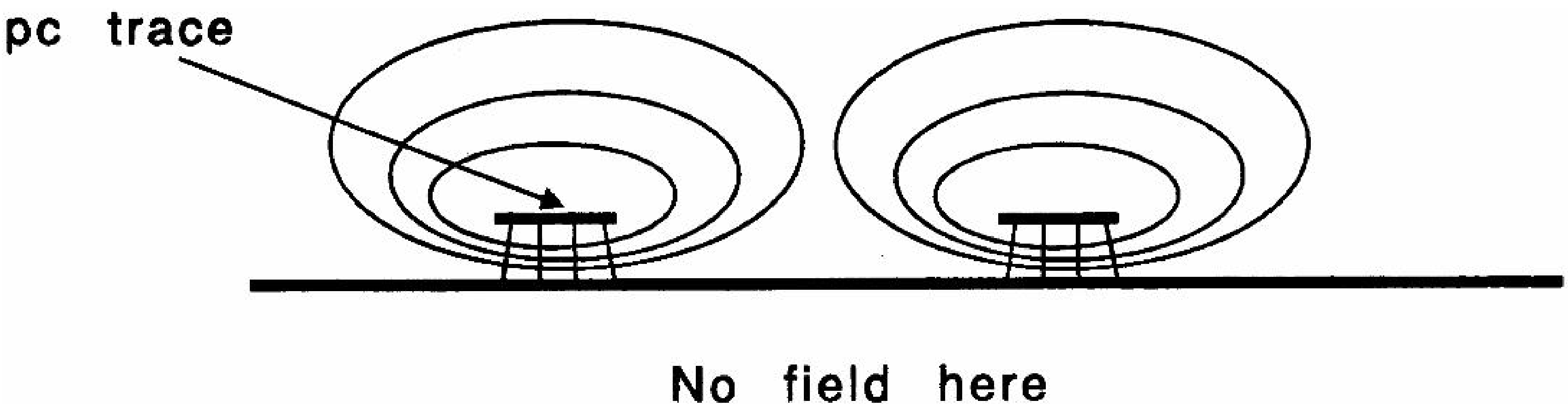
COAXIAL TRANSMISSION



Slide compliments of Ralph Morrison, Consultant

Fields are friendly

Fields concentrate under the traces and there is little crosstalk.



Fields don't penetrate the plane.

Slide compliments of Ralph Morrison, Consultant

Fields need to be carefully managed

- Every connection must be treated as part of a transmission line pair
- Field *volumes* must be carefully managed
- This is a 3 dimensional geometric design problem

Well-defined Transmission Line

Signal trace MUST be one dielectric away from the return!

Adjacent to Planar Copper
Adjacent to Ground Trace

Any deviation from this MUST be an engineered compromise, NOT an accident of signal routing

Any deviation from this WILL increase radiated emissions, degrade signal integrity, and decrease immunity

This is a very serious problem, and a big change from normal board design philosophy.

Electromagnetic Fields:

Now How do we use this wonderful information?

Where do we start?

Board outline

- Usually pre-determined
  - Defined by previous product
  - Customer requirements

Placement

- 1. Pre-defined components
  - Usually Connectors
- 2. Filter components
  - High priority, must be as close to the pins as allowed by manufacturing
- 3. Power control
  - As close to connector involved as possible
    - Voltage regulators
    - Power switching devices
    - See number 2 above

Schematic must be evaluated during layout

- Arbitrary connections can be redefined to improve layout
  - Unscrambling nets can result in:
    - Reduced complexity
    - Reduced trace length
    - Improved EMC performance
  - Signals which are not defined to specific pins
    - GPIO on MCUs
    - A/D pins on MCUs
    - Address and data lines to memories
      - No, the memory does not care what you call each pin
      - They are just address and data, not Addr14 or Data12

Schematic must be evaluated during layout

- Pin assignment to connector signals
  - Most connectors do not have adequate signal returns defined
  - Unfortunately, these are often either legacy or defined by the wiring harness
  - When possible, this can result in significant improvement in EMC behavior
  - Can have significant impact on layout complexity

Schematic must be evaluated during layout

- Pin assignment to connector signals
  - Ideal Connector Pin Assignment:

    PGSGSGSGSGP

    GSGSGSGSGSG

    – Not exactly economical or practical
  - More practical and fewer ground pins:

    SSSGSSSGSSP

    SGSSSGSSSGP

    – Each signal is still only 1 pin spacing from Ground

Schematic must be evaluated during layout

- Pin assignment to connector signals
  - Signals can be evaluated to route most critical signals adjacent to ground pins
    - Highest priority, adjacent to ground
      - labeled **A**,
    - Lower priority, diagonally adjacent to ground
      - labeled **B**,
    - Next lower priority, one pin position away from ground
      - labeled **C**
    - **…**

BAAGAAAGAAP

AGAAAGAAAGP

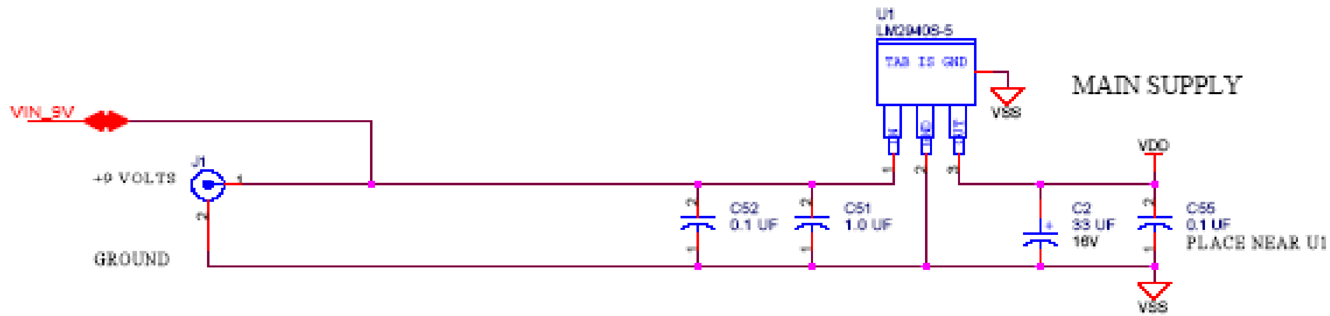
  - This can be applied when you are not allowed sufficient returns, but will improve EMC

DCAGACDCBAP

DCBABCDCAGP

Schematic must be evaluated during layout

- Schematic is often lacking in order definition
  - Capacitors must be placed in the daisy chain in the correct order



Uncontrolled component placement

- You get to decide!
- Placement not specified by customer or company requirements
- Evaluate component domain
  - Power
  - Sensor
  - Digital IC
- Place to limit signal mixing
  - Route Power only in Power realm
  - Route Sensor lines only where needed
  - Digital IC connections only in Digital realm

Uncontrolled component placement

- Power Realm devices must be placed near connectors
  - Shorter traces
  - Cleaner returns
  - Reduced field volumes
    - Yes, this is a three dimensional consideration
  - Don't forget their supporting cast
    - Bypass capacitors, Inductors, resistors, etc.
    - Use the largest value capacitor in the smallest package allowed by manufacturing and reliability [3]
- Digital Realm devices
  - Technology (geometry) of each device
  - Function
  - Devices placed within lumped distance do not need terminating resistors
    - 1/12 wavelength of the IC *switching* frequency, not clock frequency
      - Determined by IC geometry
      - Yes, this is important to know
      - Sometimes controlled by variable drive strength
    - For 1 nSec switching speeds (1 GHz) this is about ½ inch!

[3] **Comment compliments of Dr. Todd Hubing, Clemson University**

Uncontrolled component placement

- Remember, if you *do not* route signals where they don't need to be, there will not be any crosstalk or interference.
- This is easier if you *do not* mix the parts together.

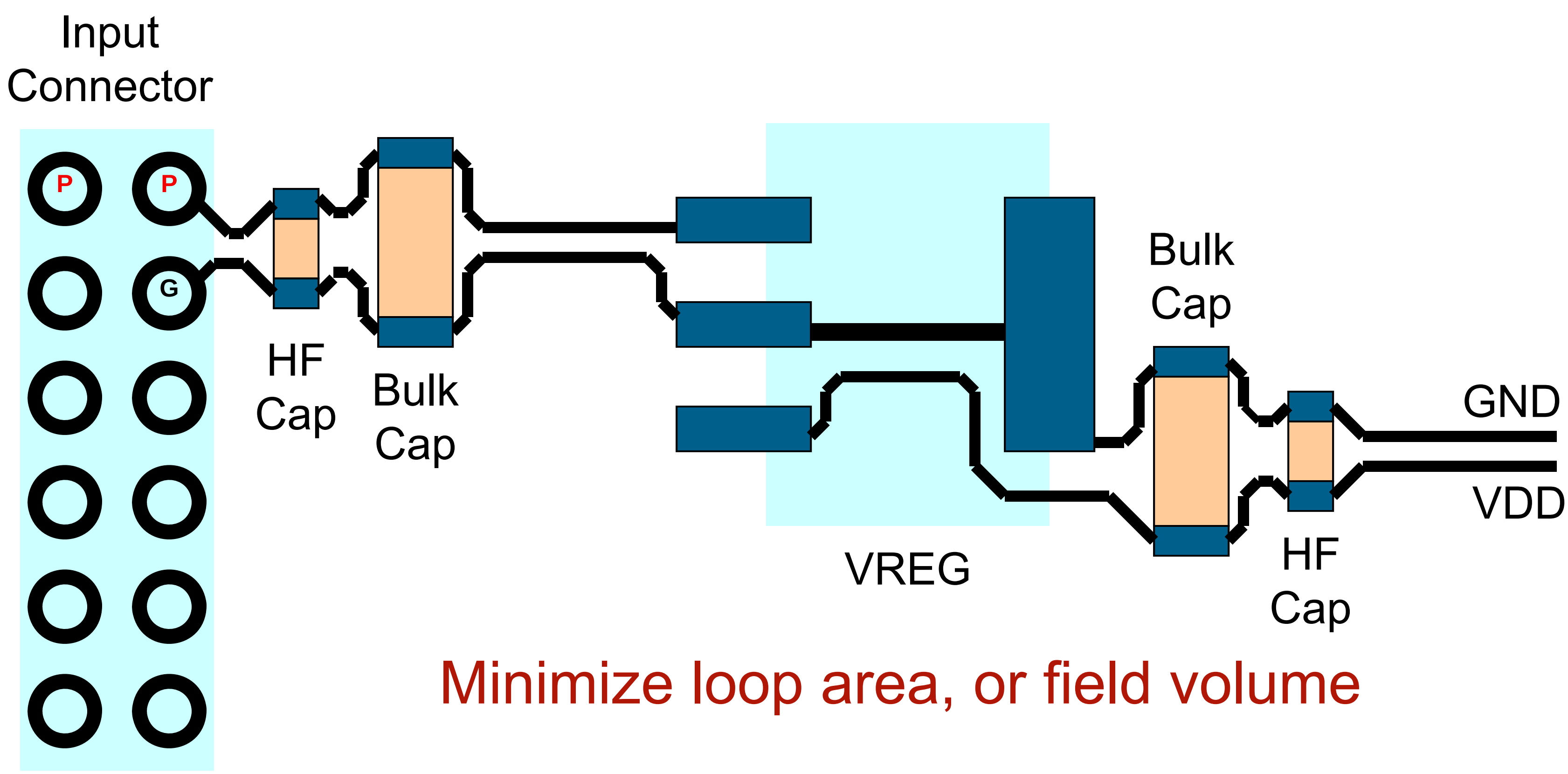
- If the traces are not near each other, there is no magic that will cause them to interfere with each other…
- Can I say this any other ways? Is this important, YES!!

Now to move on to actually routing the board…

PCB ~~Signal~~ TRANSMISSION LINE Routing

- The first and most important is to route the power distribution network, it is the source of all of the Electromagnetic energy you will be managing on the PCB.
- On low layer count boards, with no dedicated ground plane, the power lines MUST BE ROUTED IN PAIRS
  - Power *and* Ground
  - Side by Side
    - Trace width determined by current requirements
    - Spaced as close as manufacturing will allow them
  - Daisy chain from source to destination, connecting to each component, then finally to target devices
- Minimize the VOLUME of the Power TRANSMISSION network

PCB ~~Signal~~ TRANSMISSION LINE Routing



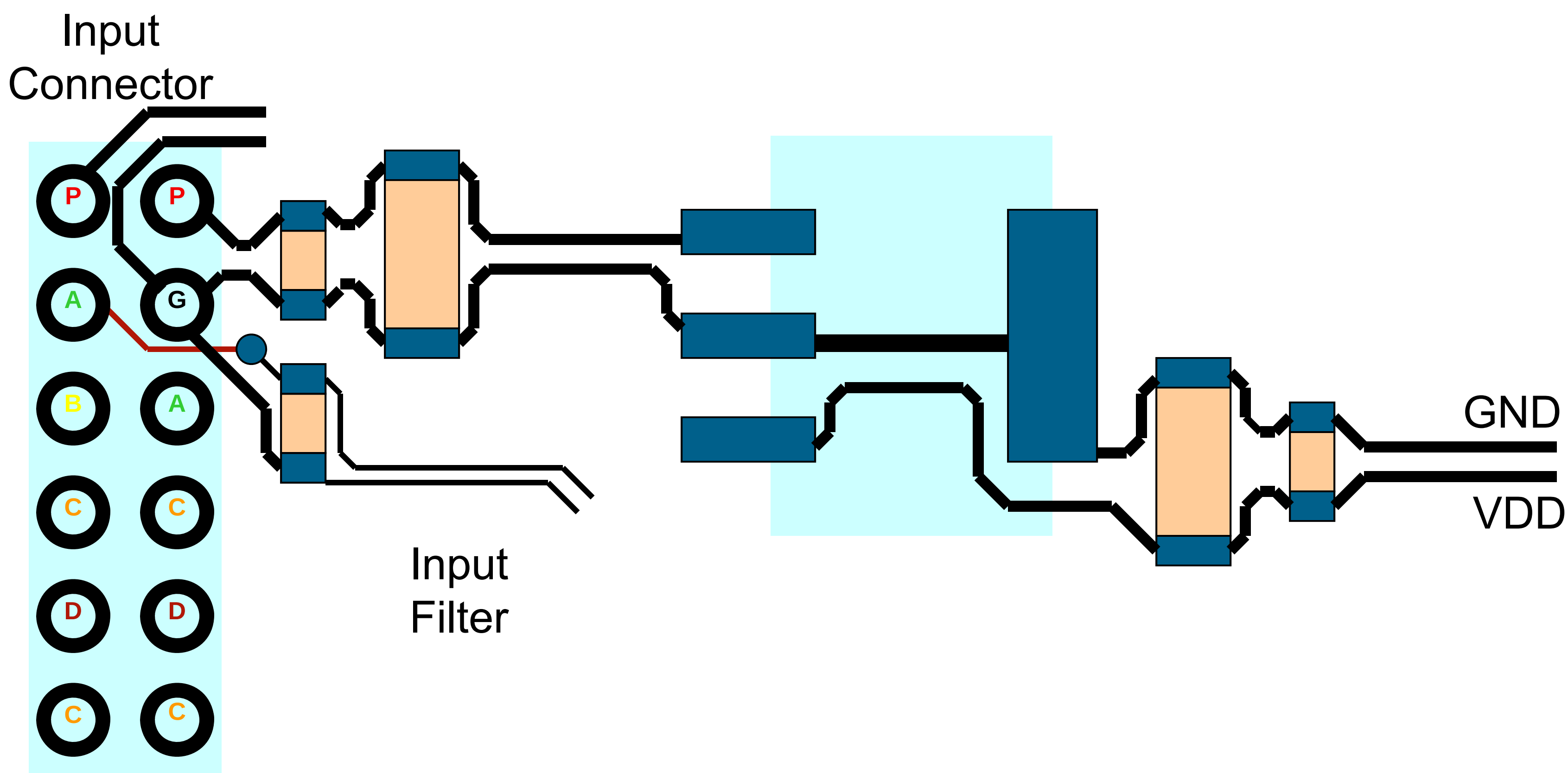
Minimize loop area, or field volume

PCB ~~Signal~~ TRANSMISSION LINE Routing

- Route power and ground traces as close as manufacturing allows
  - Internal and customer separation requirements
  - PCB Fabrication limits for chosen supplier
    - Yes, you do need to know what the supplier can manufacture
    - Can have big impact on PCB cost
- Small changes in routing can have a large impact on performance
- Component placement is critical
  - Staying within lumped distance
    - Reduces component count
    - Reduces system cost
    - Improves EMC performance

- Minimize the VOLUME of the Power TRANSMISSION network

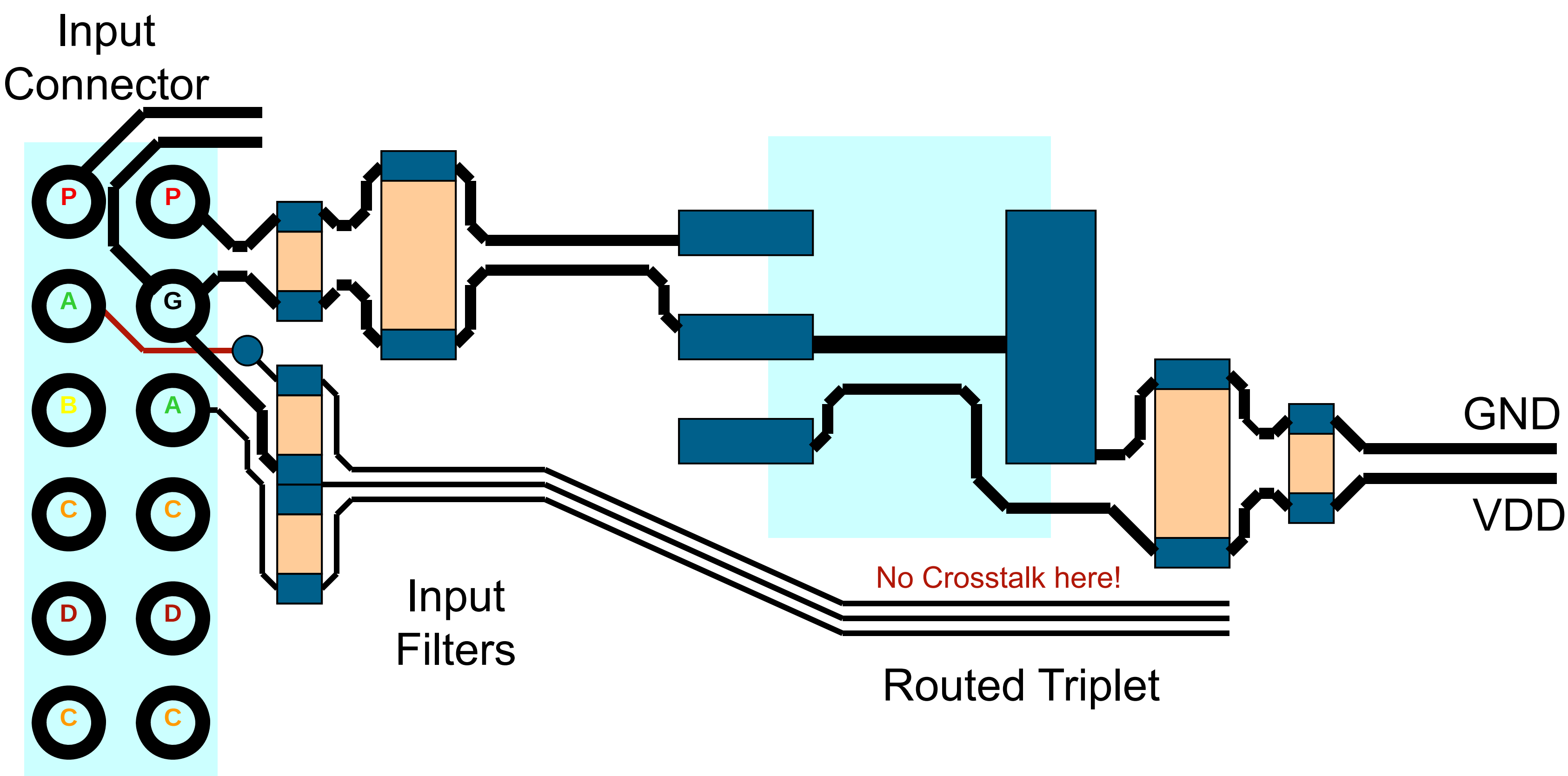
PCB ~~Signal~~ TRANSMISSION LINE Routing



PCB ~~Signal~~ TRANSMISSION LINE Routing

- Input filters must be placed as close as allowable to connectors
- Connections must be directly to the Connector Ground pins
- Route traces with well defined return path
- Minimize the VOLUME of the Signal TRANSMISSION network

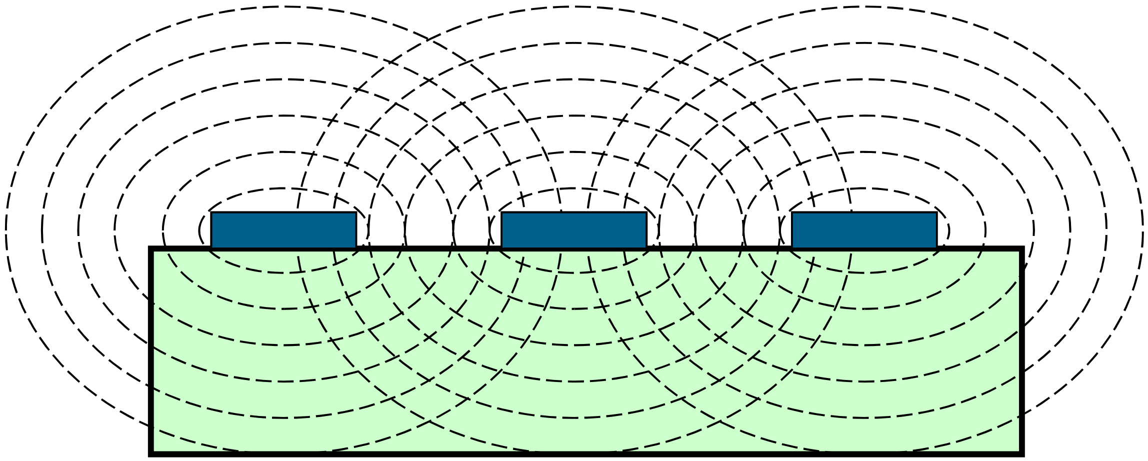
PCB ~~Signal~~ TRANSMISSION LINE Routing



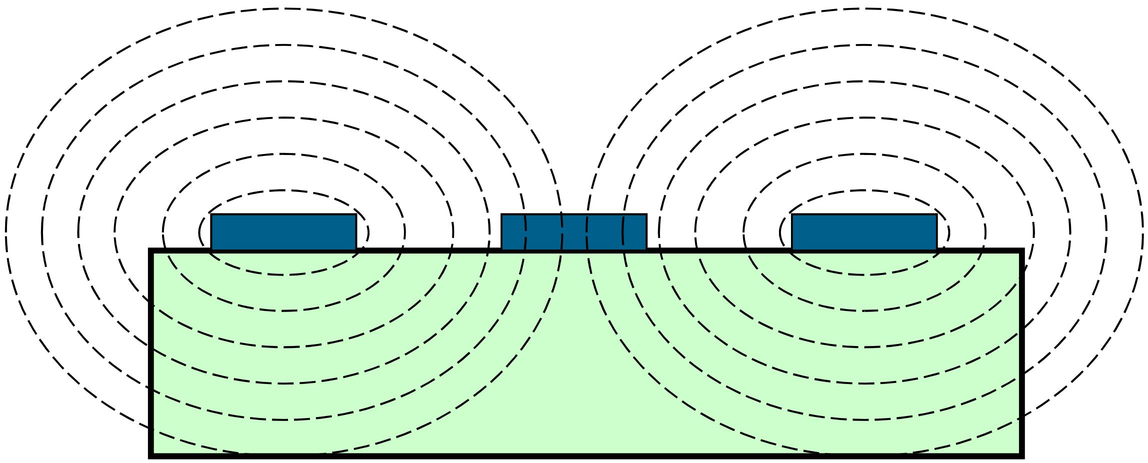
PCB ~~Signal~~ TRANSMISSION LINE Routing

- Routing in “Triplets” (S-G-S) provide good signal coupling with relatively low impact on routing density
- Ground trace needs to be connected to the ground pins on the source and destination devices for the signal traces
- Spacing should be as close as manufacturing will allow
- Minimize the VOLUME of the Signal TRANSMISSION network

You really want to make sure that the Field Energy is coupling to the conductor YOU choose!

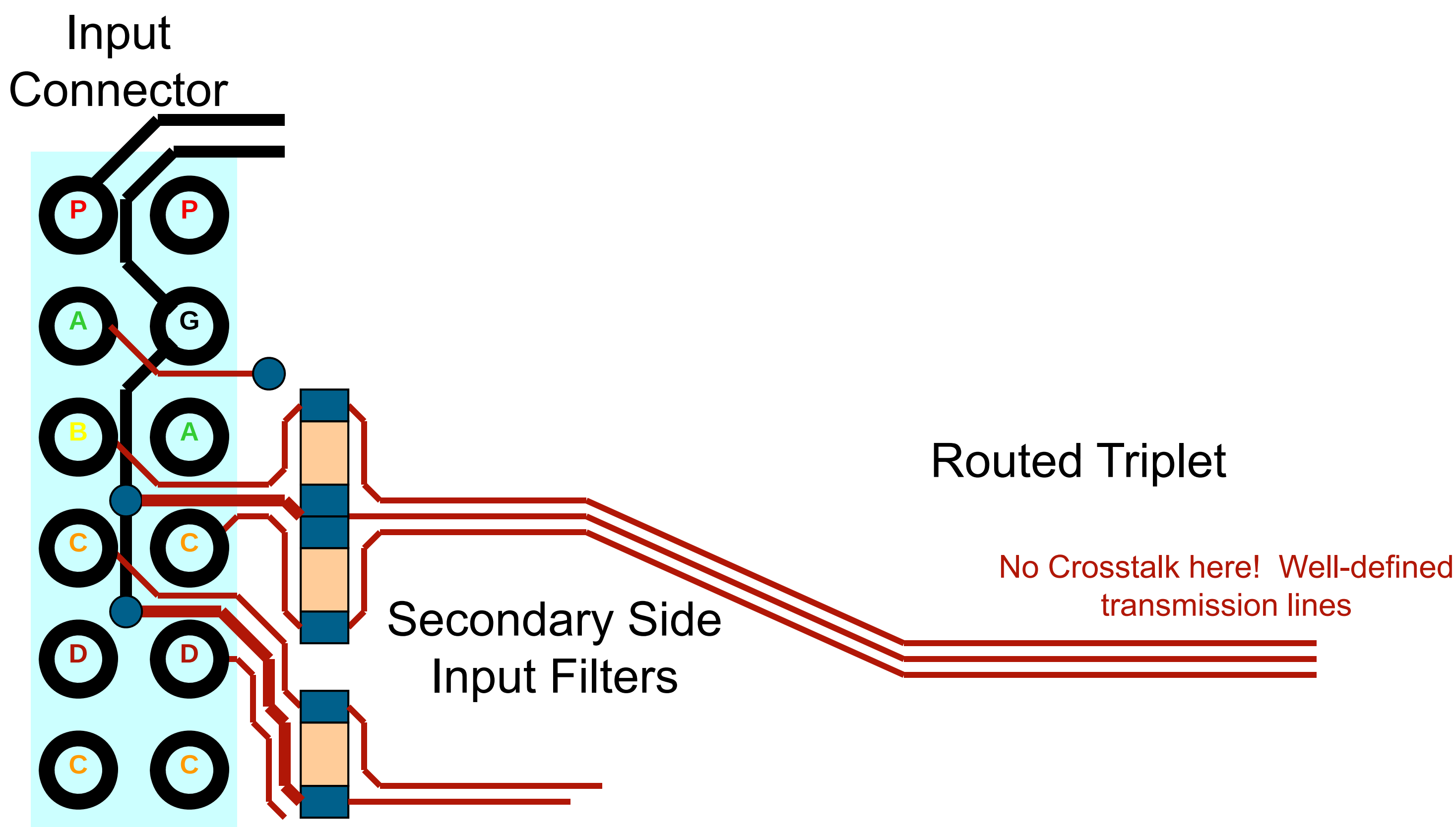


You really want to make sure that the Field Energy is coupling to the conductor YOU choose!

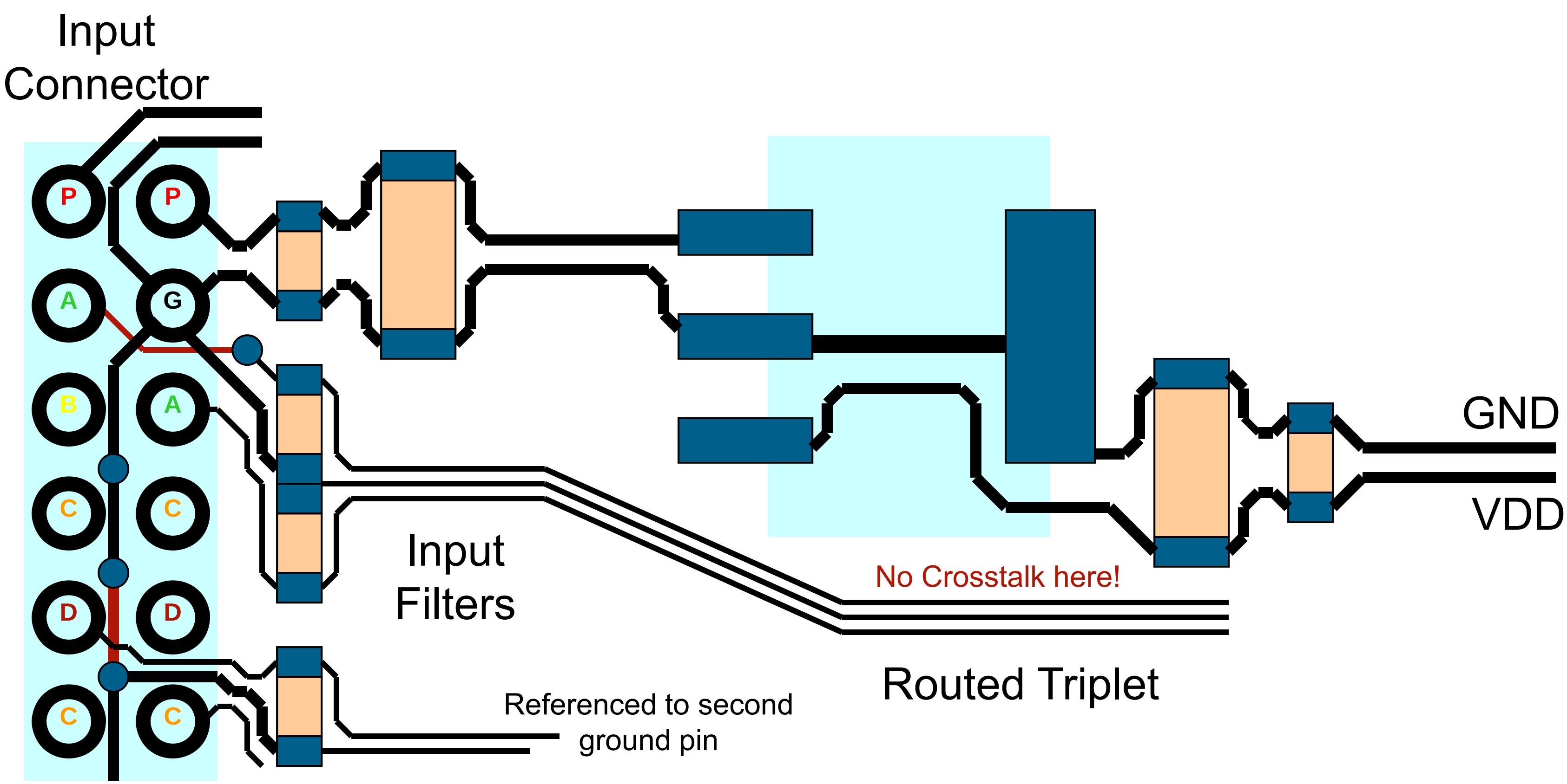


Maybe a “Triplet” makes sense???

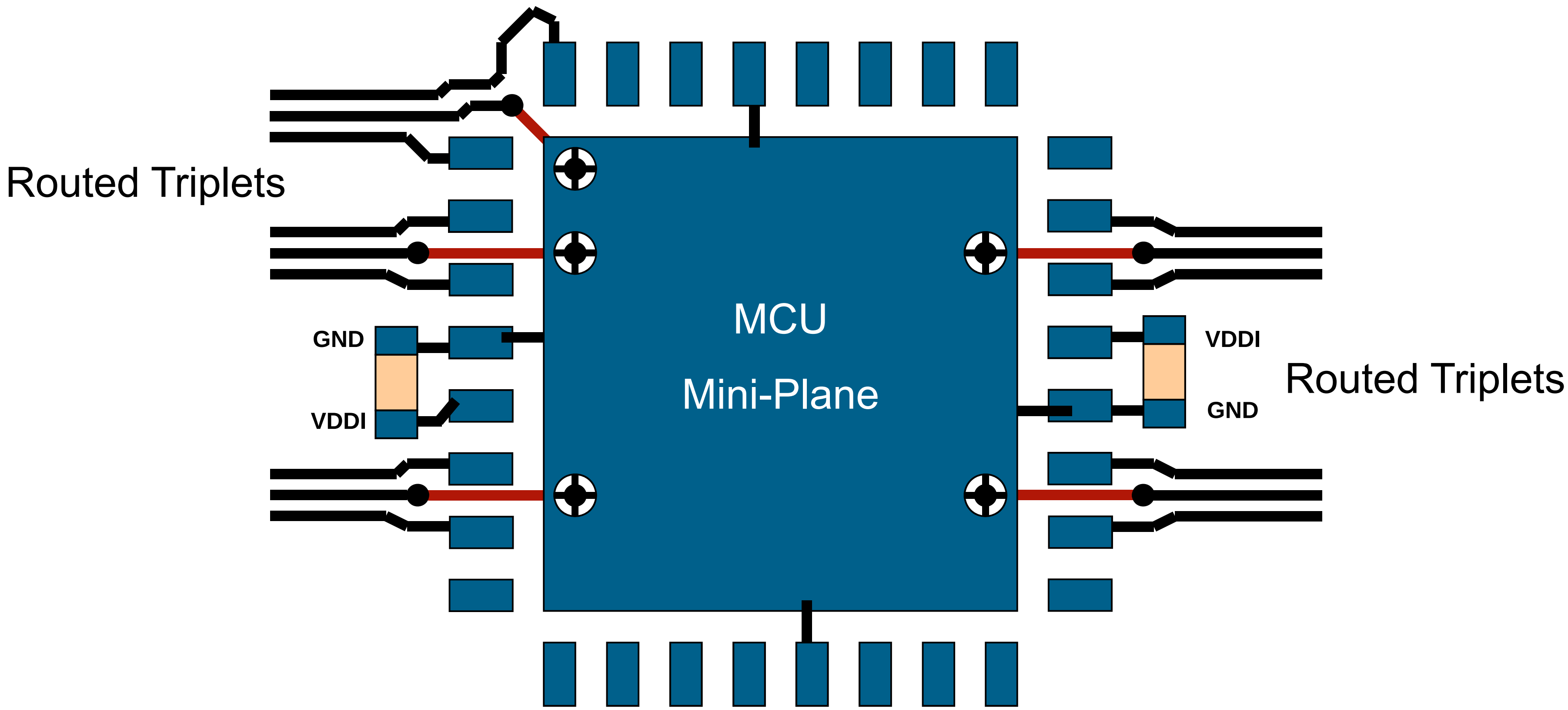
PCB ~~Signal~~ TRANSMISSION LINE Routing



PCB ~~Signal~~ TRANSMISSION LINE Routing



PCB ~~Signal~~ TRANSMISSION LINE Routing



PCB ~~Signal~~ TRANSMISSION LINE Routing

- Lead frame and wire bonds are parts of transmission lines, too.
- Mini-plane under the QFP provides improved EMC
- Triplet ground traces can be easily coupled to the Mini-plane on secondary side
- In high density applications, even routing with “Quints” (S-S-G-S-S) will provide some improvement
  - You know where most of the field energy is going!
- Last but not least, FLOOD everything with ground copper!
  - Must be able to tie each “island” with at least 2 via to adjacent layer ground

- Minimize the VOLUME of the Signal TRANSMISSION network

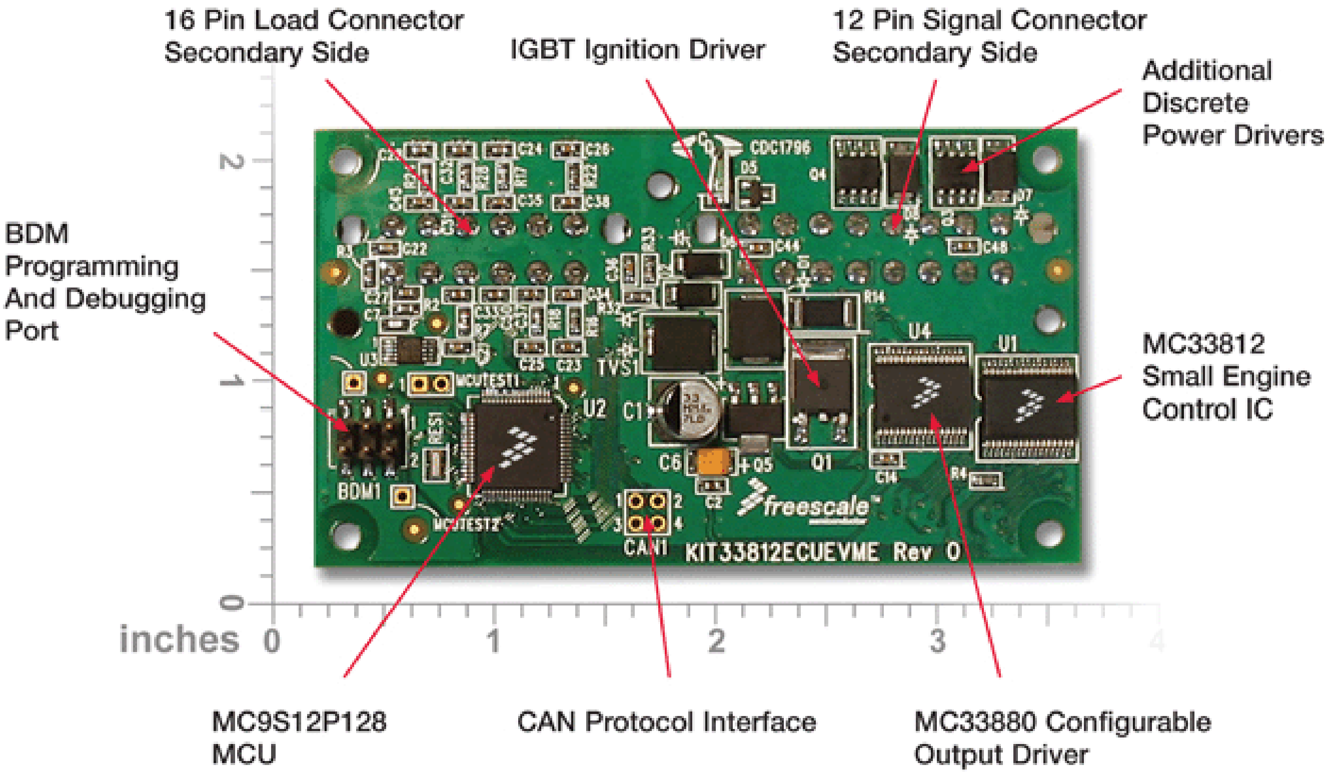
SURE, BUT DOES THIS STUFF REALLY WORK?

Testing and Evaluation

The Proof is in the Testing

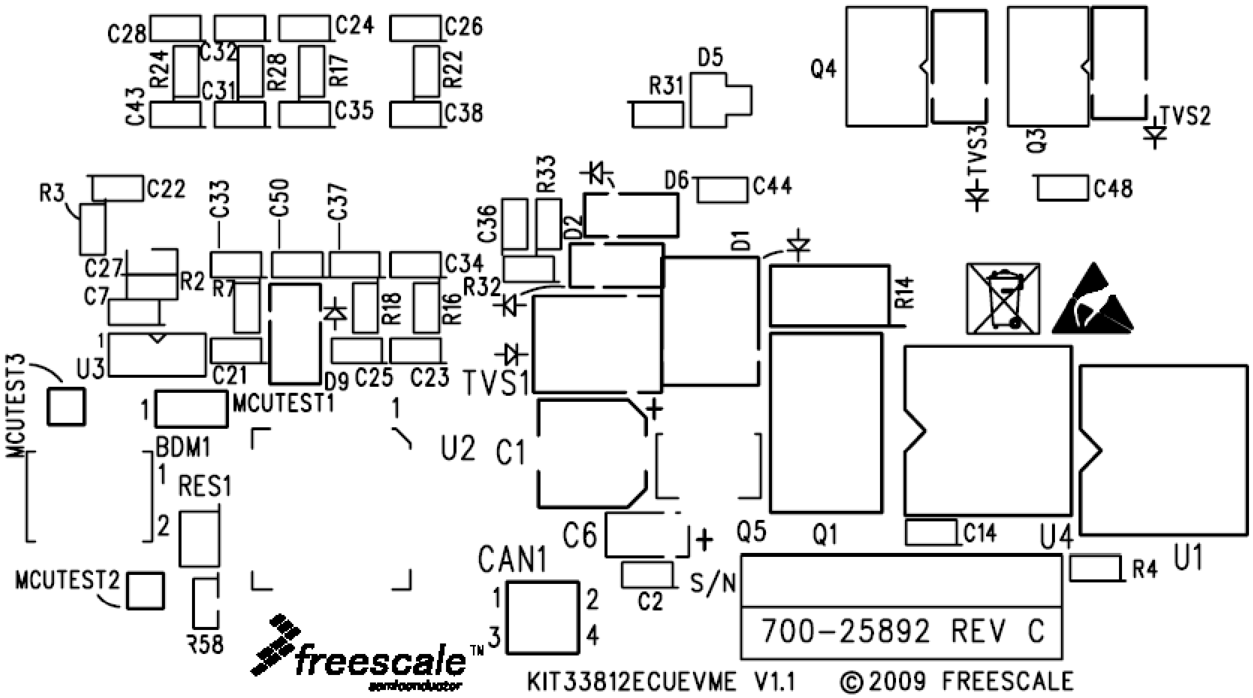
- **KIT33812ECUEVME Reference Design**
- Intended for motorcycle and other single/dual cylinder small engine control applications
- MC33812 analog power IC
  - Multifunctional Ignition and Injector Driver
- MC9S12XD128 MCU
  - Designed for either the MC9S12P128 or MC9S12XD128
  - Test results are for the older, noisier MCU
- Two Layer PCB
- Business Card dimensions
- Implements these Design and layout concepts
  - “Smart” connector pinout
  - MCU Mini-Plane
  - Triplet routing
  - Maximum Flooding

KIT33812ECUEVME Reference Design



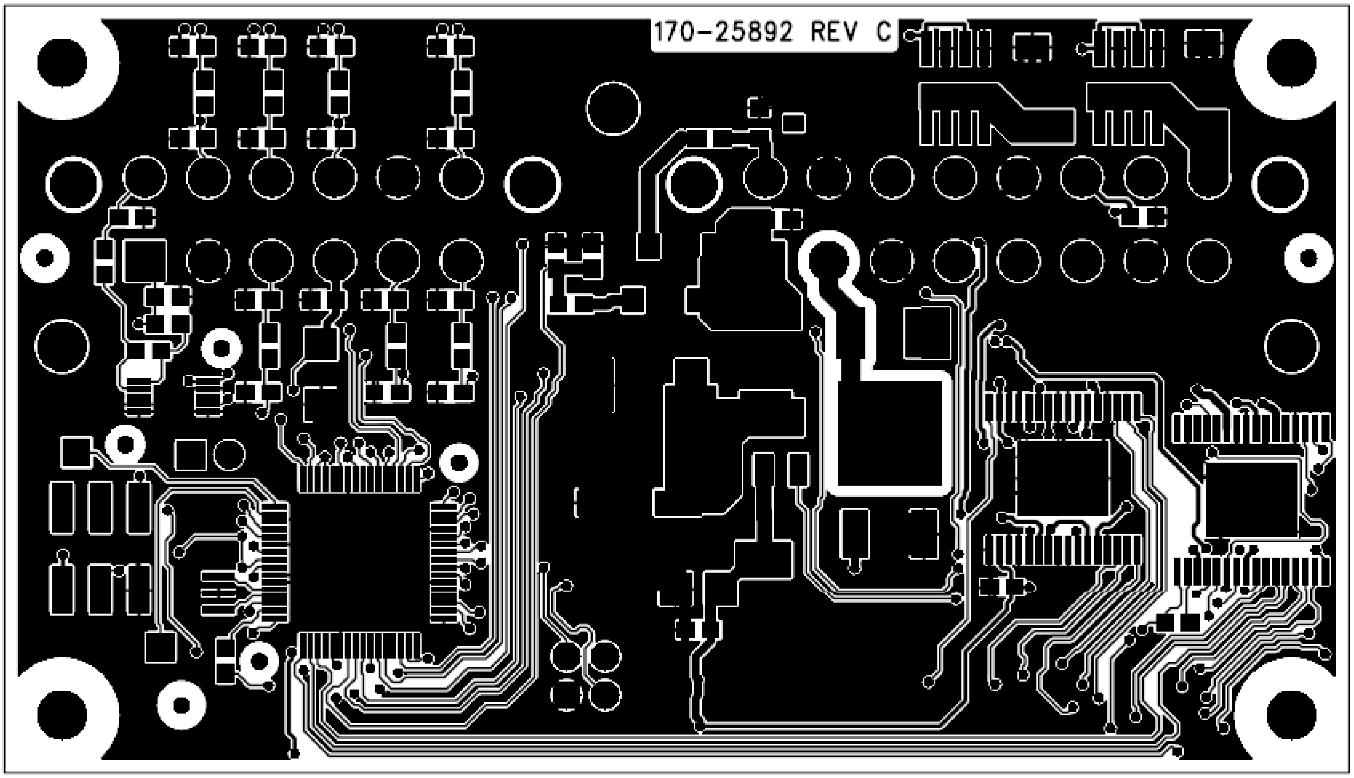
KIT33812ECUEVME Reference Design

Primary Silk



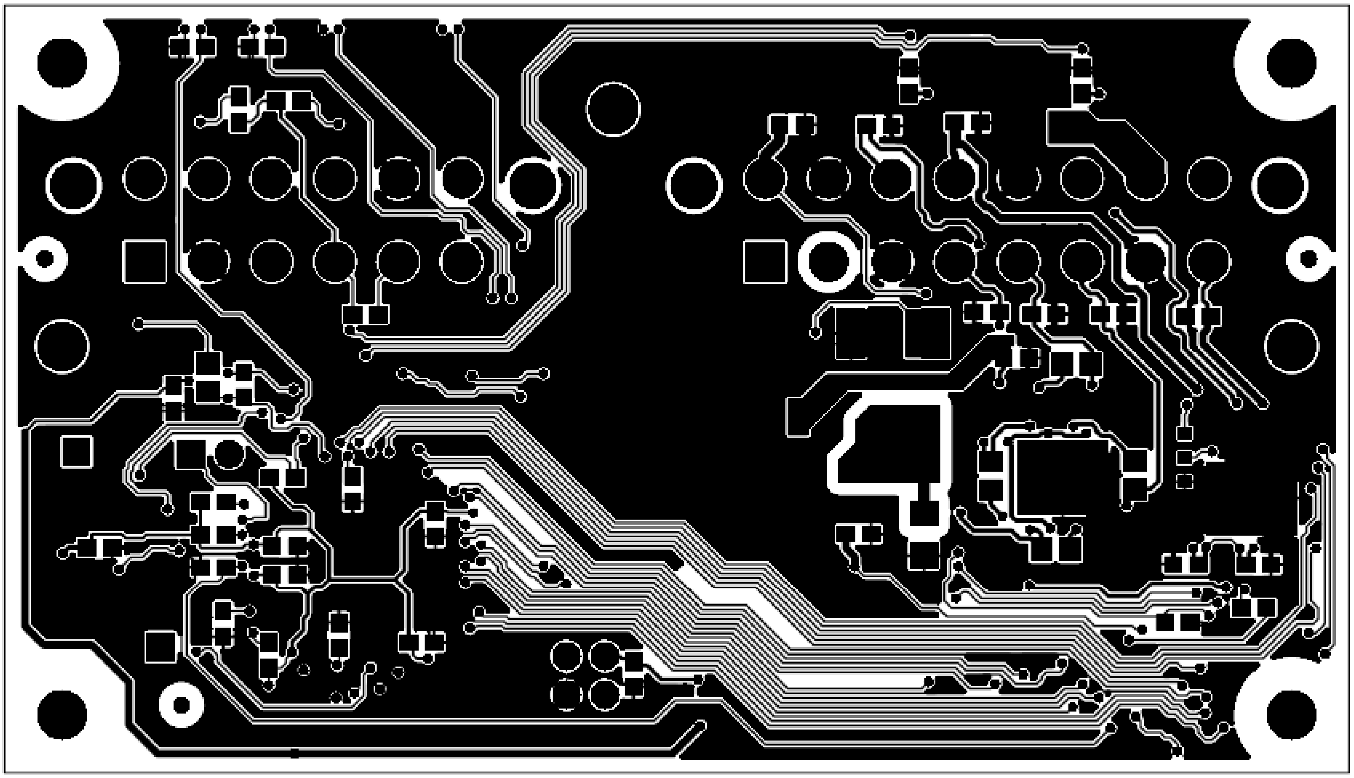
KIT33812ECUEVME Reference Design

Primary Side

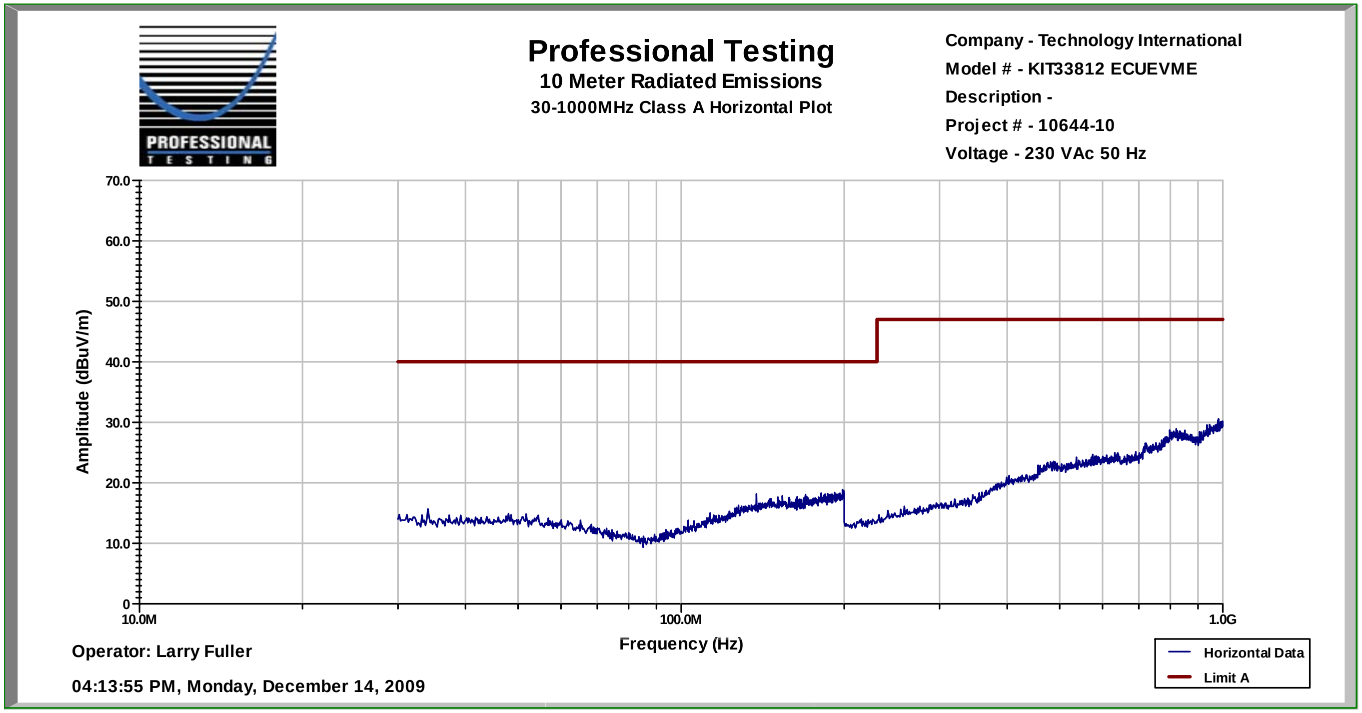


KIT33812ECUEVME Reference Design

Secondary Side

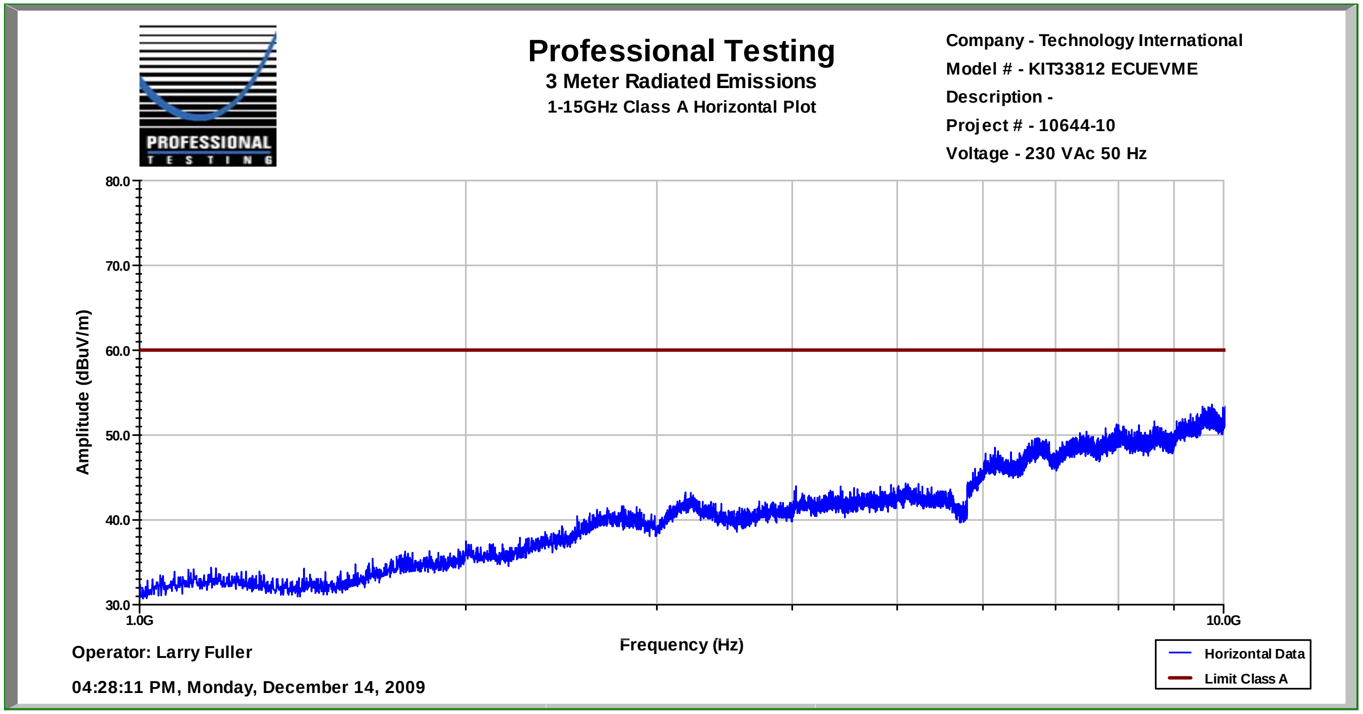


KIT33812ECUEVME Reference Design



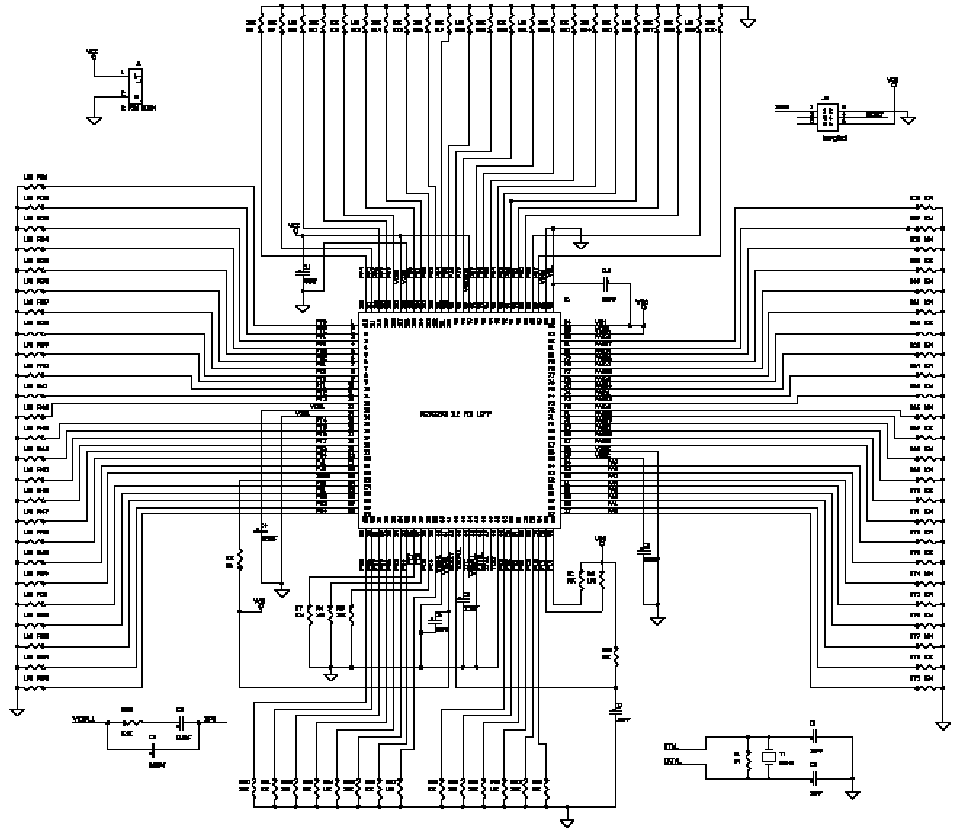
Effective PCB Design:
Techniques to improve performance

KIT33812ECUEVME Reference Design



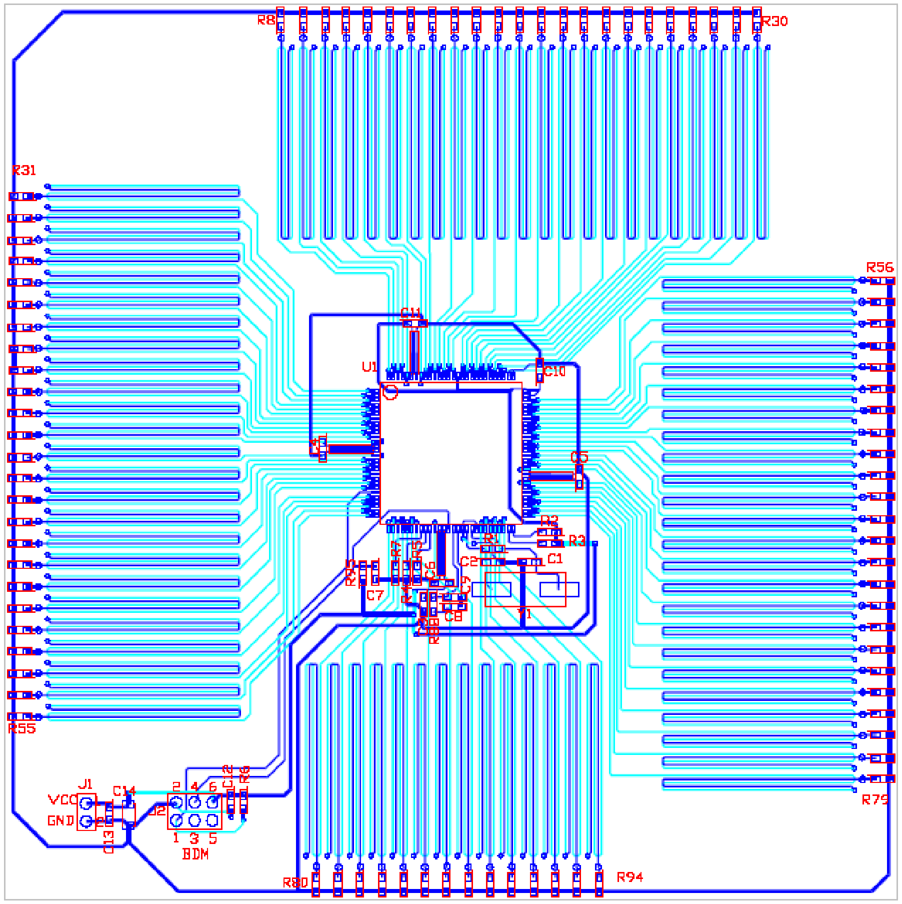
EMC Test Board

- EMC test board with no Field control considered
- Two layers
- 112 pin MC9S12XD128 MCU
- All I/O lines routed to 10 K termination resistors using serpentine 6” traces
- All ground connections routed in “convenient” patterns
- Filter components placed “somewhere near”
- Line widths and spacing aimed for low cost FAB
- Software running at 40 MHz, toggling all I/O pins



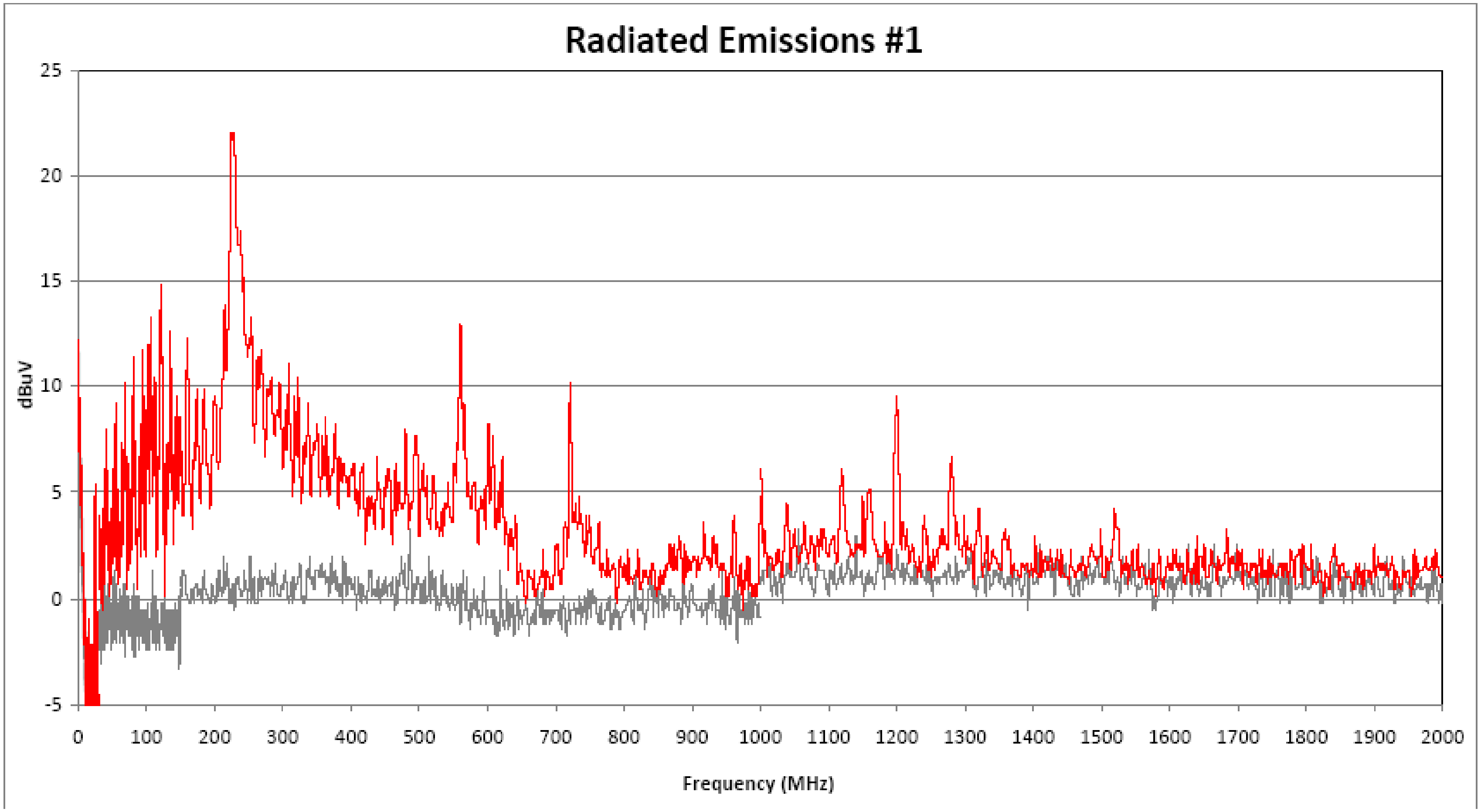
EMC Test Board Schematic

Effective PCB Design:
Techniques to improve performance



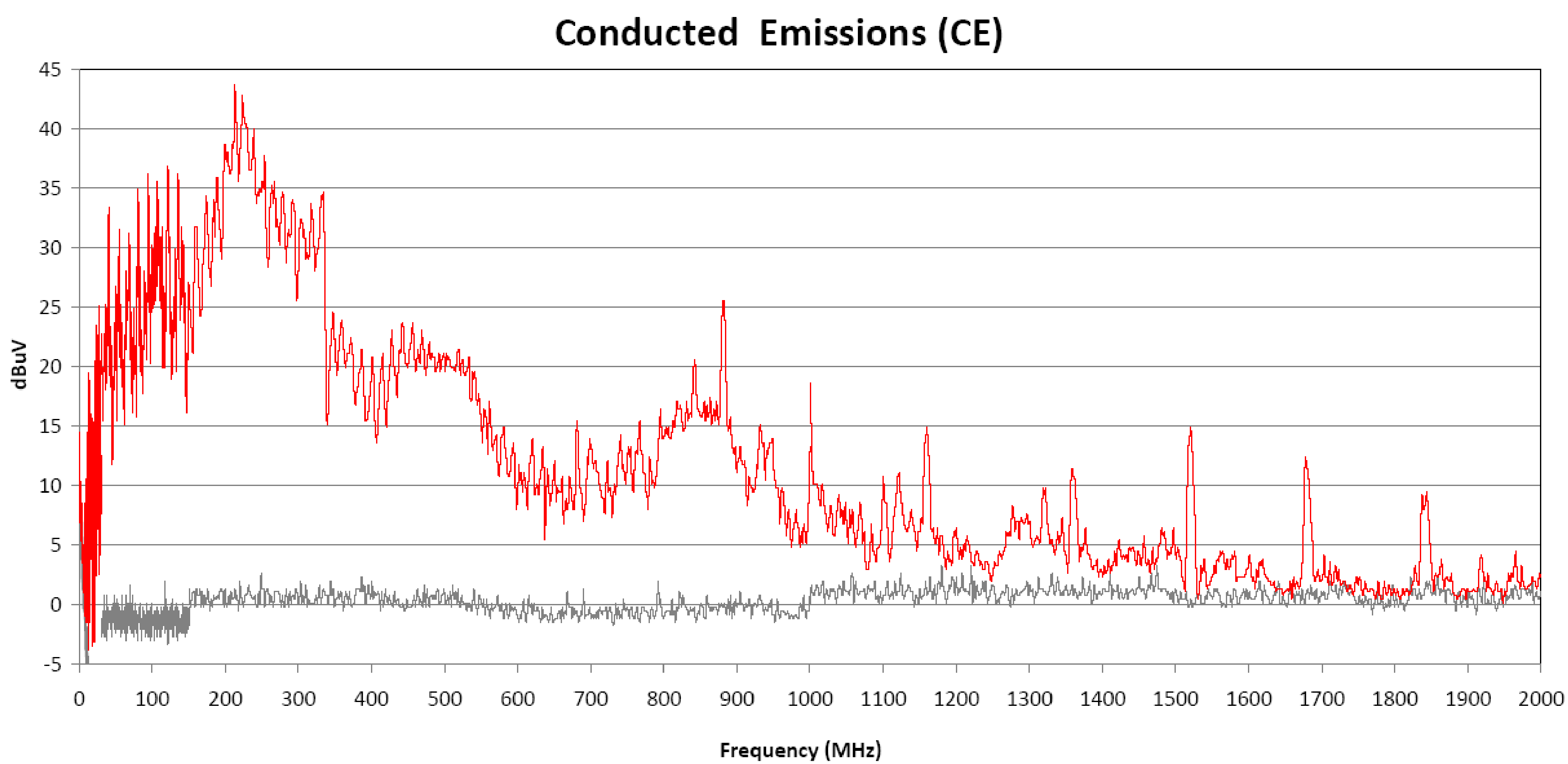
EMC Test Board Layout

Effective PCB Design:
Techniques to improve performance



2 Layer EMC Test Board Radiated Emissions

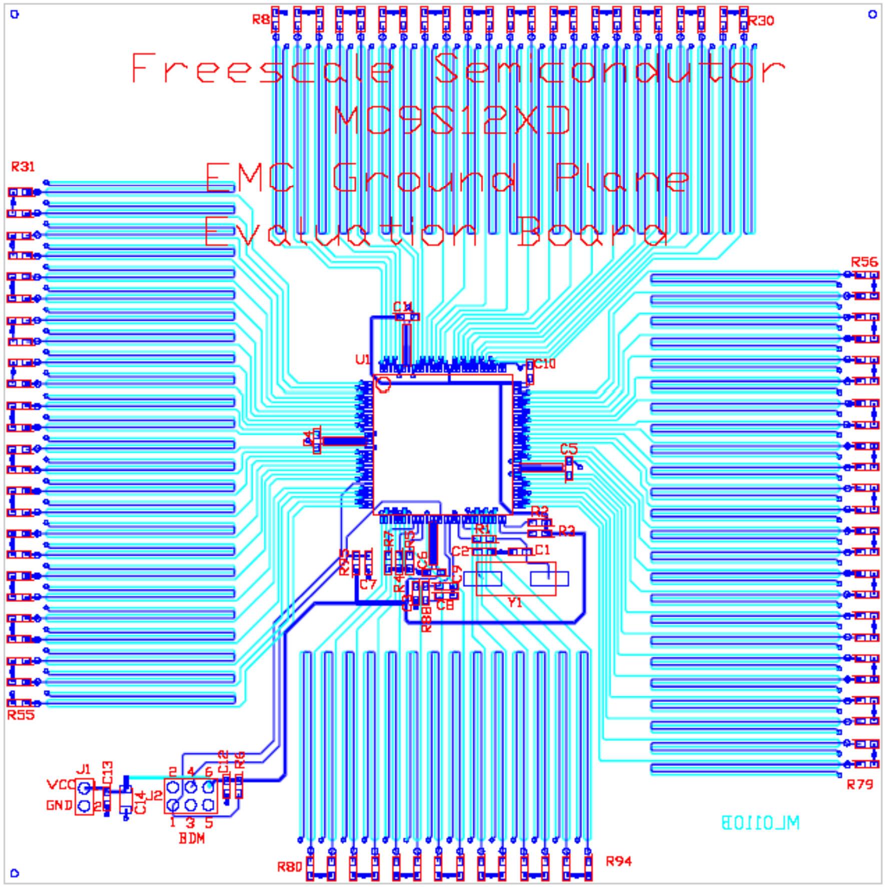
Effective PCB Design:
Techniques to improve performance



2 Layer EMC Test Board Conducted Emissions

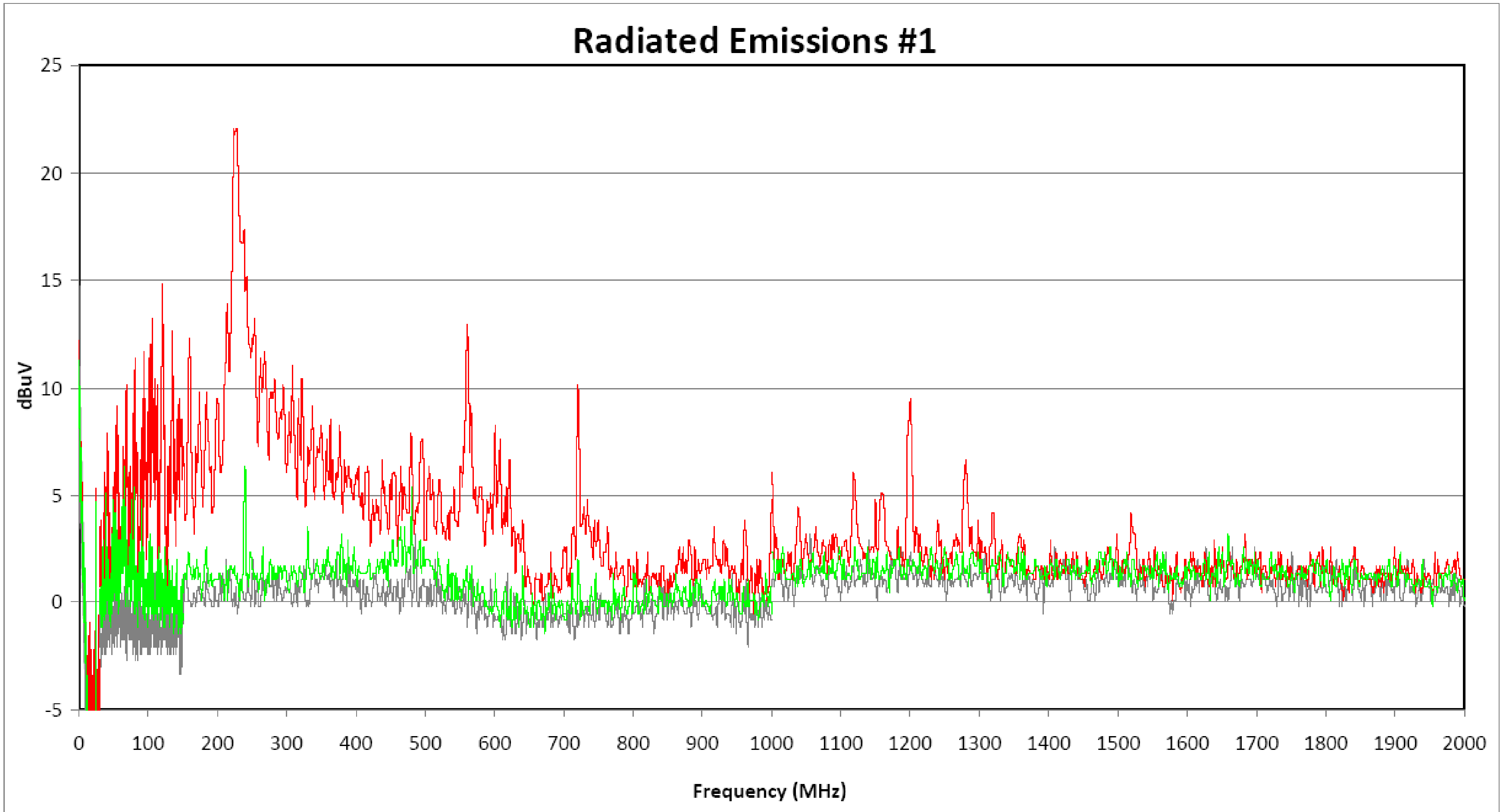
EMC Test Board, rev 2

- EMC test board with Tight Field control considered
- Same schematic
- Four layers
  - Core inserted with dedicated Ground Planes
- Outer layers exactly the same as 2 layer
- All ground connections made with via to ground planes
- Line widths and spacing aimed for low cost FAB
- Same software

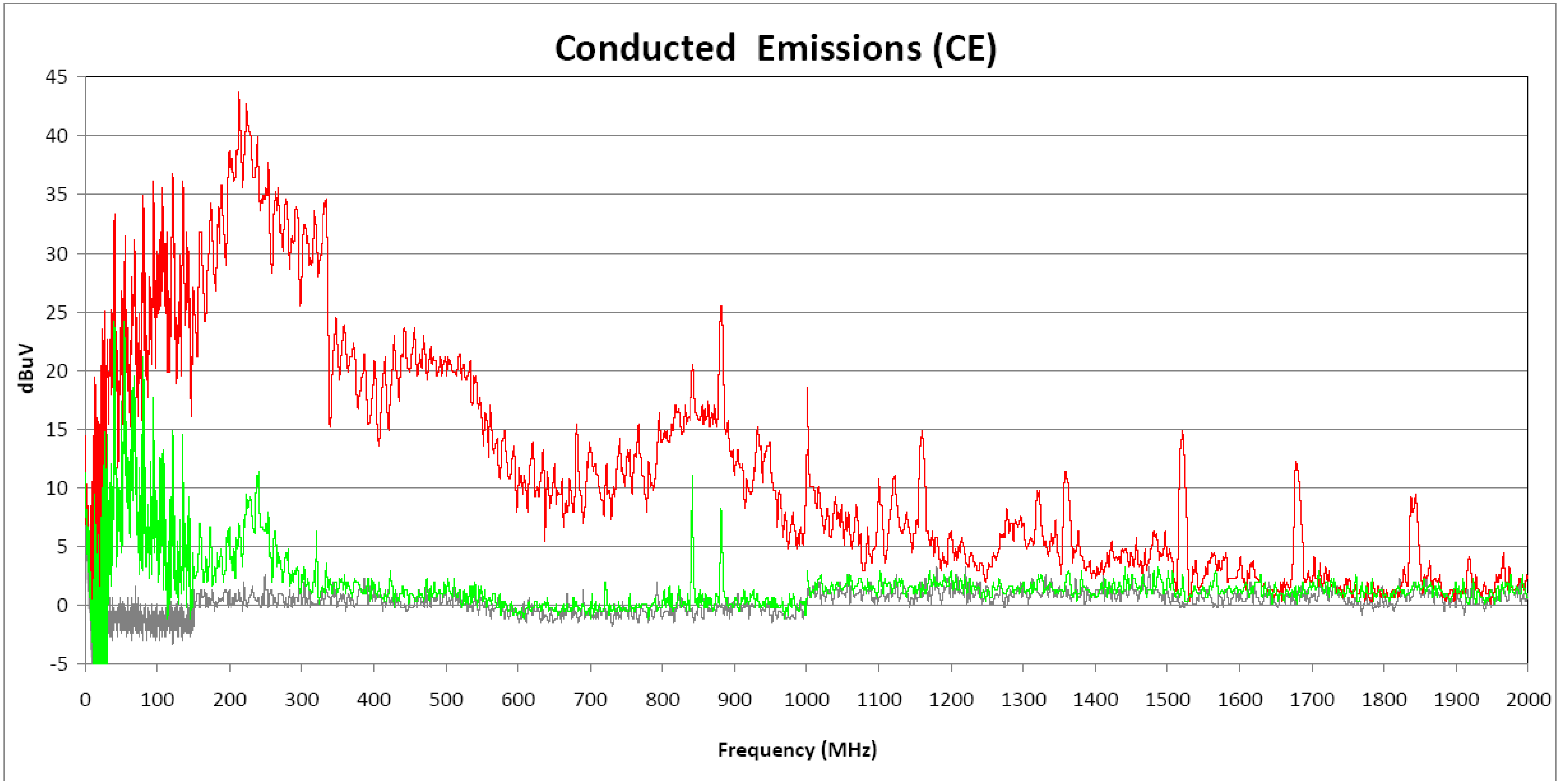


EMC Test Board Layout

Effective PCB Design:
Techniques to improve performance



2 VS 4 Layer EMC Test Board Radiated Emissions



2 VS 4 Layer EMC Test Board Conducted Emissions

WOW!

30 db of Improvement!

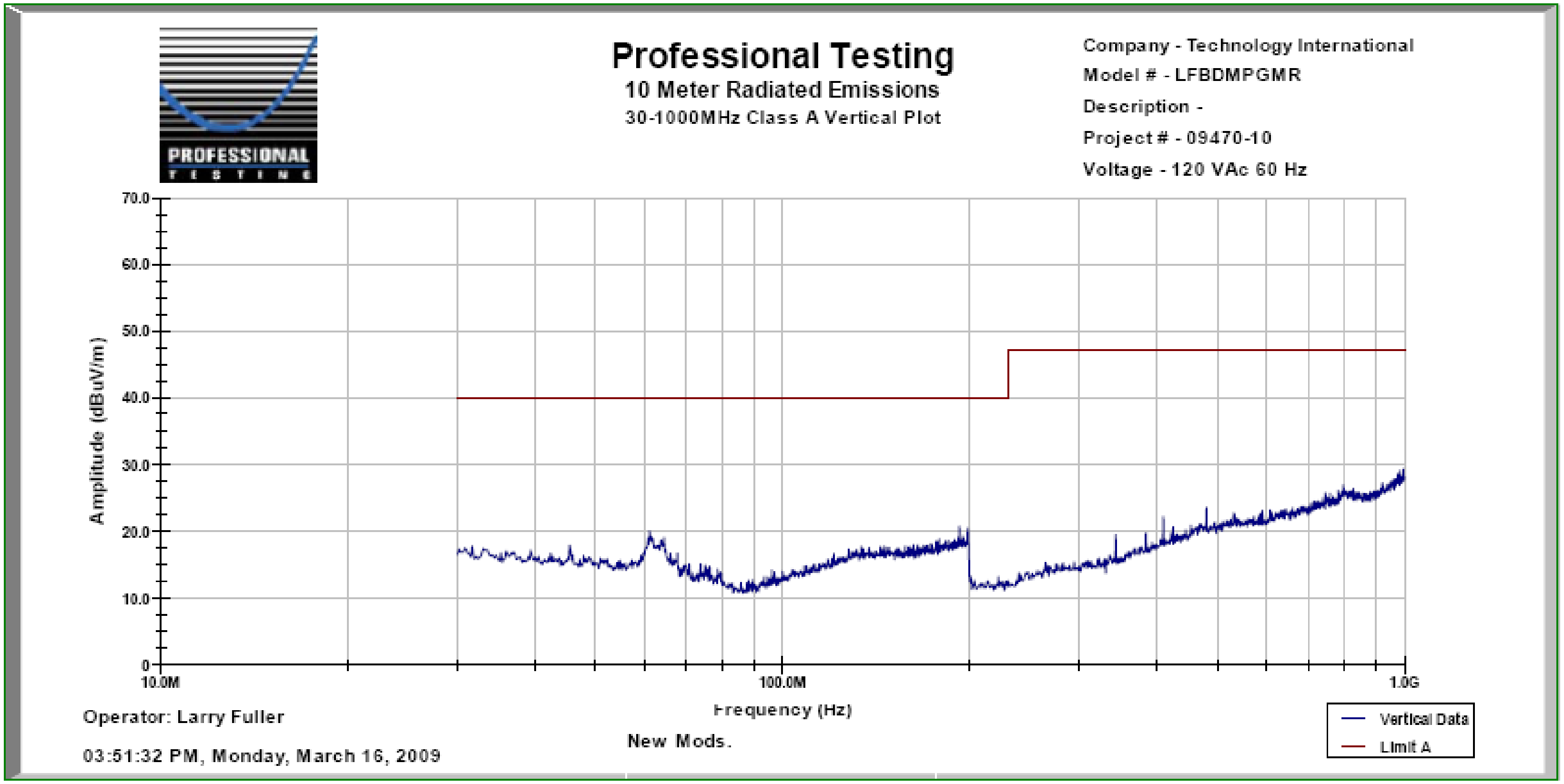
EMC Test Results

- EMC test results can be used to identify area of concern
- LFBDMPGMR FCC/CE test result first pass:
  - Radiated Immunity
  - "The EUT failed with all led's turning off. Manual restart worked. The frequencies that caused this fault were 110 MHz, 112 MHz, 134 MHz, and 136 MHz up to 149 MHz. After 149 MHz the EUT worked properly."
- Not what you want to see in your email
- This is a 4 layer board, the best I know how to design!!??
- I know, check the chart to see what the ¼ wave length would be
- About 1 meter, what? My board is only 4 inches square.
- Aha, the USB cable!! I forgot to put a filter on the USB power supply. Add a cap quick.
- Send new board for retest

Antenna size vs. Frequency

Frequency	¼ wave length
1 Hertz Rise time equivalent, who cares	246,000,000 feet (46,591 miles) Almost 6 times around the earth
10 Hertz Rise time equivalent, still who cares	24,600,000 feet (4,659 miles) Almost from New York to Honolulu
100 Hertz Rise time equivalent, .01 seconds	2,460,000 feet (466 miles) Almost from New York to Detroit
1 KHz Rise time equivalent, 1 millisecond	246,000 feet (46.6 miles) Almost from Orlando to Cocoa Beach
10 KHz Rise time equivalent, 100 microseconds	24,600 feet (4.659 miles) Almost from the J. W. Marriott to Disney's Magic Kingdom
100 KHz Rise time equivalent, 10 microseconds	2,460 feet (0.466 miles) Almost from the J. W. Marriott to the Central Florida Parkway
1 MHz Rise time equivalent, 1 microsecond	246 feet (0.0466 miles) Less than a football field
10 MHz Rise time equivalent, 100 nanoseconds Rise time distance, 100 feet	24.6 feet Across the room
100 MHz (TTL Logic) Rise time equivalent, 10 nanoseconds Rise time distance, 10 feet	**2.46 feet Less than a yard**
1 GHz (BiCMOS Logic) Rise time equivalent, 1 nanosecond Rise time distance, 1 foot	0.246 feet (2.952 inches) Less than your finger
10 GHz (GaAs Logic) Rise time equivalent, 100 picoseconds Rise time distance, 1.2 inches	0.0246 feet (0.2952 inches) Less than the diameter of a pencil
100 GHz (nanometer geometry HCMOS) Rise time equivalent, 10 picoseconds Rise time distance, 0.12 inches	0.00246 feet (0.0295 inches) Half the thickness of a standard FR4 PCB

EMC Test Results



EMC Test Results, Yeah!!

EN 61000-4-3
Radiated Immunity
Technology International
LFBDMPGMR

Test Date: March 13, 2009	Client: Technology International
Project #: 09470-10	Supervisor: Jason Anderson
EUT: LFBDMPGMR	Technician: Dan Keenan

EUT Power Source: 120VAC
Ambient Temperature: 22.6 ° C
Barometric Pressure: 29.97 inches
Relative Humidity: 55 %

EUT Face Illuminated	Frequency Range							
	80-200 MHz		200-1000 MHz		1.4-2.0 GHz		2.0-2.7 GHz	
	3 V/m		3 V/m		V/m		V/m	
	Horizontal	Vertical	Horizontal	Vertical	Horizontal	Vertical	Horizontal	Vertical
Front	X	X	X	X				
Right	X	X	X	X				
Rear	X	X	X	X				
Left	X	X	X	X				

Test Results: Pass [X] Fail [ ]

The EUT met performance criteria:

Criteria A	X
Criteria B	
Criteria C	
Manufacturers Specification	

Notes: The RF signal was modulated with 80% 1000 Hz modulation. The frequency step size was 1% of the preceding frequency. The dwell time at each frequency was 2 seconds.

PCB Layout Considerations

Some new "Rules of Thumb"

More PC Board Considerations

- Flooding unused spaces on the PCB
  - Properly implemented will improve EMC performance
  - Reduce cost by increasing PCB manufacturing yield
    - Less etch required
    - Balanced copper improves plating
    - Balanced copper improves final assembly
      - Reduced board warping

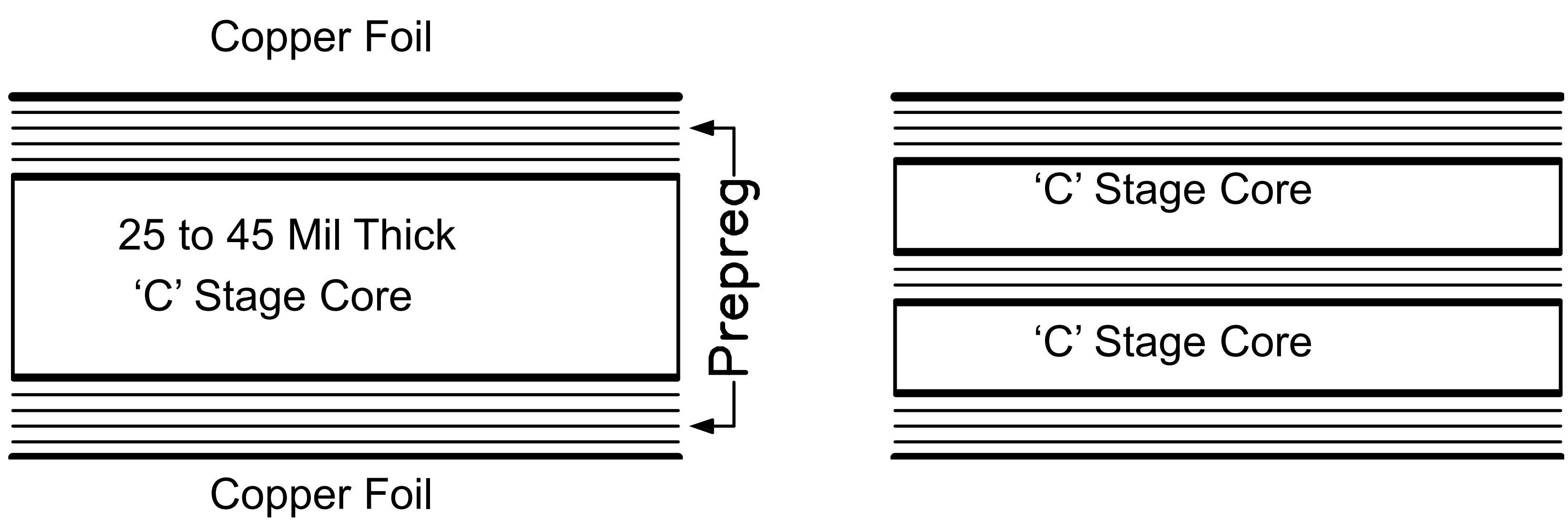
More PC Board Considerations

- Use the minimum trace widths and spacing for signal transmission lines
  - Refer to PCB fabricator’s capabilities without a cost adder
  - May be defined by either customer or internal requirements
  - Wider traces for power supply transmission line pairs
  - Provides maximum trace density
- Makes room for all of those ground traces!

More PC Board Considerations

- Four layer boards
  - Made from a 2 layer core, L2 and L3
  - L1 and L4 made by adding pre-preg layers and copper foil
  - Use the "fattest" core and "thinnest" pre-preg possible without a cost adder from fabricator
    - You will have to find this out
    - Your company or customer may have some min-max specs for these materials
  - Maximum coupling is from L1 to L2 and from L3 to L4

Effective PCB Design:
Techniques to improve performance



<u>Most PC Boards are "Foil Laminated"</u>

Slide compliments of Rick Hartley, Consultant

More PC Board Considerations

- Layer count determinations
  - Technology of the devices used
  - Trace density
  - EMC certification level
    - Consumer/Commercial
    - Automotive
    - Aviation
    - Military, etc.

All must be considered, not just Trace Density!!!

More PC Board Considerations

- Layer count determinations
  - Must be a conscious decision based on proper electromagnetic field control
  - Not just because you ran out of routing paths
  - Smaller IC geometries will require more layers and most likely power and ground planes
  - It will not be possible to provide a good power distribution network or good signal integrity without adding planes

System cost is ***NOT*** reduced by reducing IC geometries!!

More PC Board Considerations

I repeat:

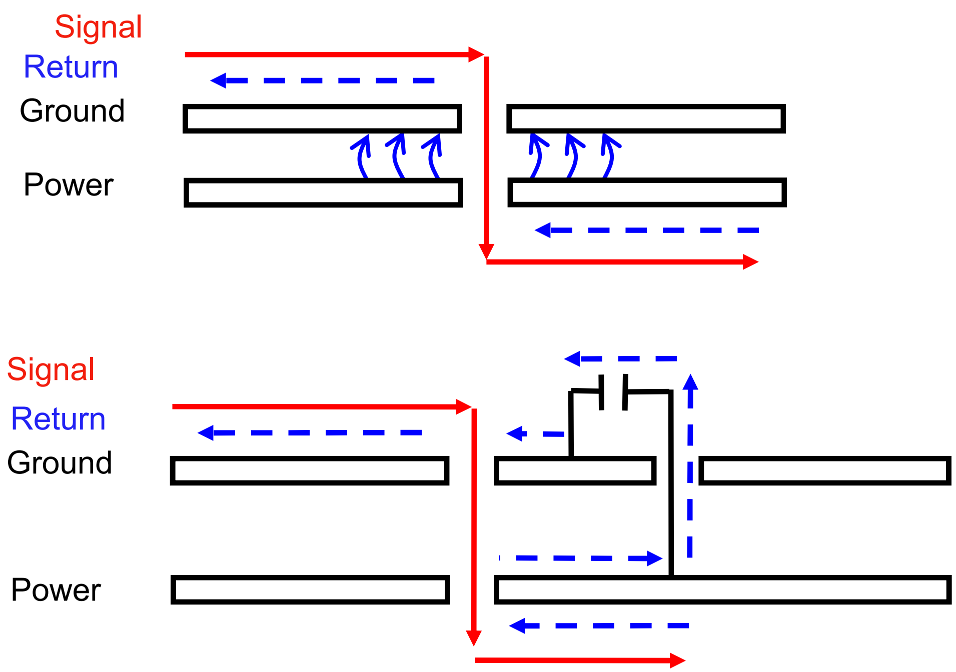
System cost is ***NOT*** reduced by reducing IC geometries!!

More PC Board Considerations

- Using Planes
  - Both Power and Ground can be used as signal references
    - ONLY if they are well coupled to each other
      - ►Capacitors
      - ►Adjacent to each other
  - Transition from one reference plane to another requires close proximity to a bypass capacitor
    - That is the only way the energy can go!

Effective PCB Design:
Techniques to improve performance

When routing signals with returns between Power and Ground Planes, Return energy will transfer as follows -

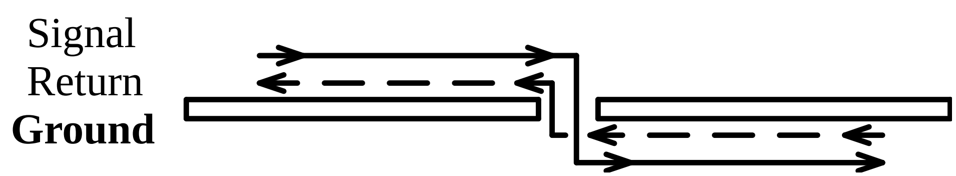


Tightly Coupled Planes

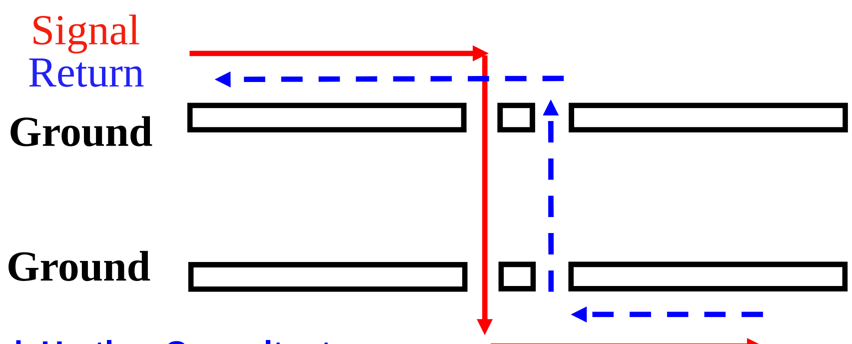
Loosely Coupled Planes w/ Cap

Slide compliments of Rick Hartley, Consultant

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



When moving signals between 2 different planes, use a transfer via VERY near the signal via.



Slide compliments of Rick Hartley, Consultant

More PC Board Considerations

- Remember, Field energy moves in the space between or around the conductors and cannot go through them [1]
  - That means through the holes in the planes
    - Not inside or on the vias, around them!!
- You must provide the path you want, or the field will find its own path
  - It will most likely be the one that causes the most problems!

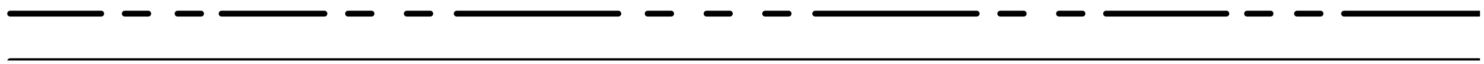
[1] **Statement compliments of Ralph Morrison, Consultant**

More PC Board Considerations

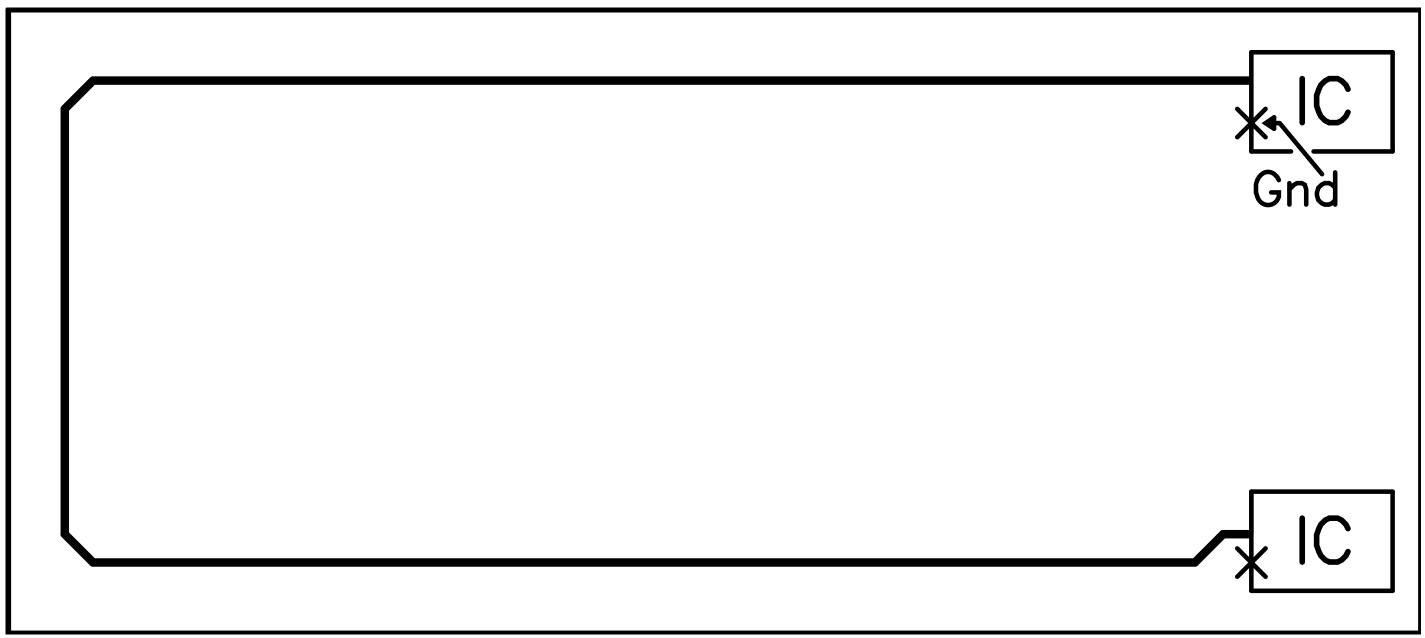
- Using Planes
  - Splitting Ground Planes is almost never a good idea
    - Only when required by customer or internal specifications
    - Question those requirements!!
  - If you have to split a plane, do not route traces across the split!
    - If you must, then you absolutely have to route a following ground trace across the split next to the signal trace

Splits in Planes are very efficient Slot Antennas!!

2 Layer Microwave Style PC Board -



L2- Ground.

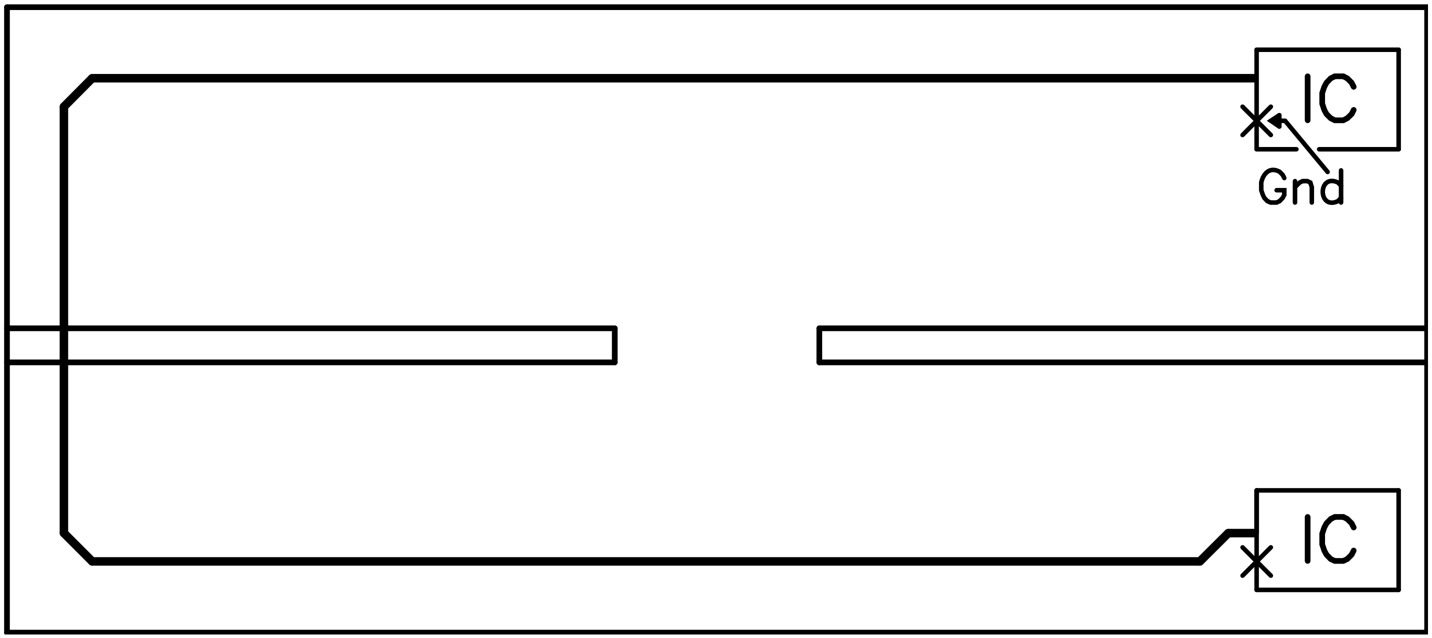


Where does signal's return current flow?

Slide compliments of Rick Hartley, Consultant

What happens if Return Plane is Split???

- Now where does return current flow?

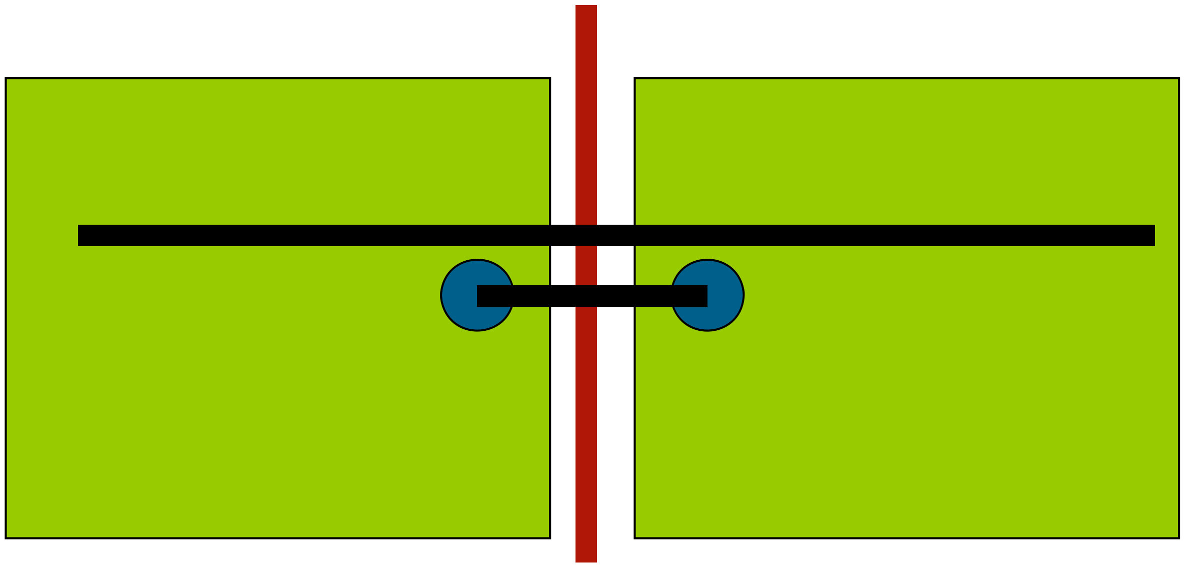


Where does signal's return current flow?

Slide compliments of Rick Hartley, Consultant

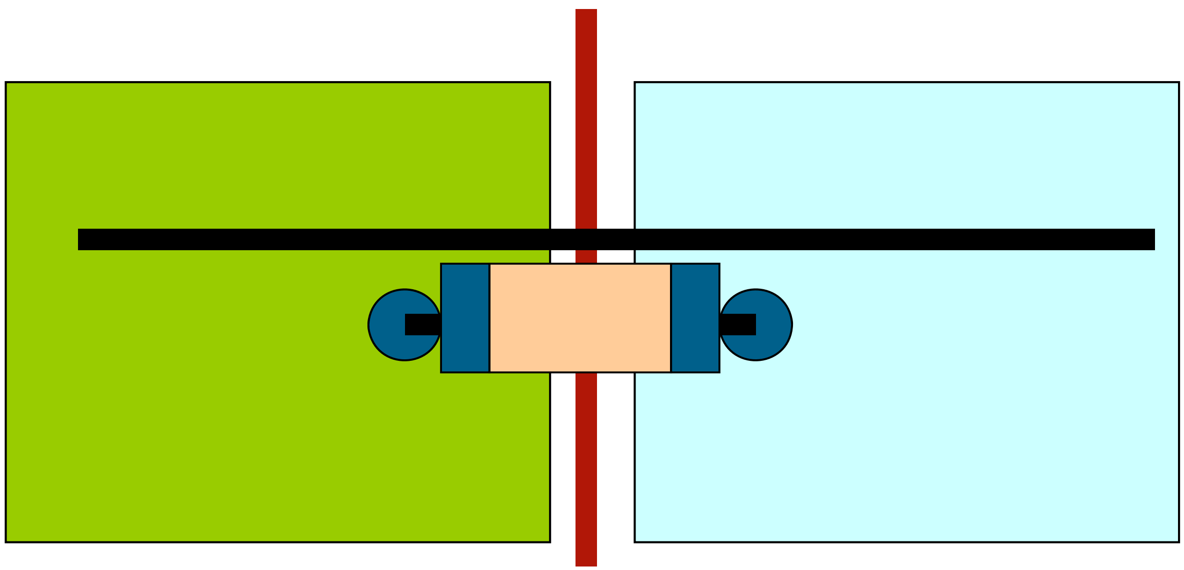
More PC Board Considerations

- Routing over Split Planes, Same Potential
  - Just use a bridge tied to each plane
  - Better to just not split it, but sometimes you have to route a trace in the split



More PC Board Considerations

- Routing over Split Planes, Different Potential
  - Have to bridge with a capacitor



More PC Board Considerations

- Routing Differential signals
  - Myth: They are coupled to each other
  - Fact: They are coupled to Ground
  - They do not have to be routed together
  - They do need to be about the same length
  - They do need to be treated as transmission lines
    - You knew I was going to say that, didn't you?
  - They would benefit from being routed as a "Triplet"
  - Designed to reject Common Mode noise

More PC Board Considerations

- Routing Timing Critical Bus Signals
  - Myth: They have to be exactly the same length
    - Manufacturers often spec allowable trace length differential
    - PCB designers spend a lot of time and energy to do this using serpentines and other extreme routing methods
    - At high frequency, the serpentines are invisible anyway, and actually result in SHORTER travel times
  - Fact: What matters is the set up and hold time required by the devices
    - This is usually specified in time, i.e. ps
    - Remember this? **v = 150 mm / ns or 6" / ns**
- For a typical 500 MHz DDR memory interface, the data lines *only* need to be within 500 mils of each other in length [2]
  - Way easier than we have been led to believe

[2] **Statement compliments of Rick Hartley, Consultant**

Closing Remarks and Reference Materials

PCB Design is not a Black Art!

- Well defined transmission lines result in significantly improved EMC performance
- Careful routing of transmission lines can result in behavior similar to that gained by adding extra PCB ground layers
- Evaluating test results can lead you to solutions
- The Black Magic is tamed!

Effective PCB Design:

Techniques to improve performance

My special thanks and accolades to my patient and extremely tolerant mentors:

- Rick Hartley, PCB designer extraordinaire, who started me down this trail in 2004 at PCB West.
- Ralph Morrison, Author, Inventor, and Musician, who has patiently and steadily moved me from the fuzzy realm of "Circuit Theory" and "Black Magic" into the solid world of physics.
- Dr. Todd Hubing, Researcher and Professor, whose research at UMR and Clemson have provided solid evidence that Maxwell and Ralph have got it right!

Finally, My team here at Freescale, we have really come a long way!

High Speed Design Reading List

1. Right the First Time- A Practical Handbook on High Speed PCB and System Design - Volumes I & II - Lee W. Ritchey (Speeding Edge) - ISBN 0-9741936-0-7
2. High Speed Digital System Design- A handbook of Interconnect Theory and Practice - Hall, Hall and McCall (Wiley Interscience 2000) - ISBN 0-36090-2
3. High Speed Digital Design- A Handbook of Black Magic - Howard W. Johnson & Martin Graham (Prentice Hall) - ISBN 0-13-395724-1
4. High Speed Signal Propagation- Advanced Black Magic - Howard W. Johnson & Martin Graham - (Prentice Hall) - ISBN 0-13-084408-X
5. Signal Integrity Simplified - Eric Bogatin (Prentice Hall) - ISBN 0-13-066946-6
6. Signal Integrity Issues and Printed Circuit Design - Doug Brooks (Prentice Hall) - ISBN 0-13-141884-X

[2] **Slide compliments of Rick Hartley, Consultant**

EMI Reading List

1. PCB Design for Real-World EMI Control - Bruce R. Archambeault (Kluwer Academic Publishers Group) - ISBN 1-4020-7130-2
2. Digital Design for Interference Specifications- A Practical Handbook for EMI Suppression - David L. Terrell & R. Kenneth Keenan (Newnes Publishing) - ISBN 0-7506-7282-X
3. Noise Reduction Techniques in Electronic Systems - Henry Ott (2nd Edition - John Wiley and Sons) - ISBN 0-471-85068-3
4. Introduction to Electromagnetic Compatibility - Clayton R. Paul (John Wiley and Sons) - ISBN 0-471-54927-4
5. EMC for Product Engineers - Tim Williams (Newnes Publishing) - ISBN 0-7506-2466-3
6. Grounding & Shielding Techniques - Ralph Morrison (5th Edition - John Wiley & Sons) - ISBN 0-471-24518-6

[2] **Slide compliments of Rick Hartley, Consultant**

Some additional references you may find useful:

http://www.ralphmorrison.com/Ralph_Morrison/Welcome.html
Ralph Morrison's website

http://pcbwest.com/
Best PCB design conference website

http://www.emcesd.com/
Doug Smith's website (He is the best at finding what is wrong! Lots of useful apnotes.)

http://www.emcs.org/
IEEE EMC Society website

http://www.cvel.clemson.edu/auto
Clemson's Automotive Electronics website

http://www.cvel.clemson.edu/emc
Clemson's EMC website

http://www.mst.edu/about/
Missouri University of Science and Technology website

http://www.ipc.org/default.aspx
IPC — Association Connecting Electronics Industries website

"Buildings have walls and halls.

*People travel in the **halls** not the walls.*

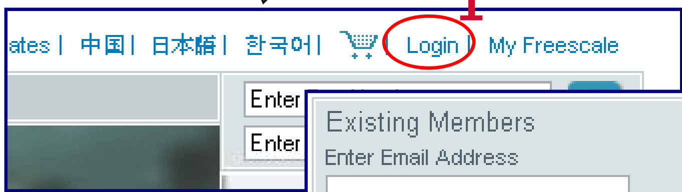
Circuits have traces and spaces.

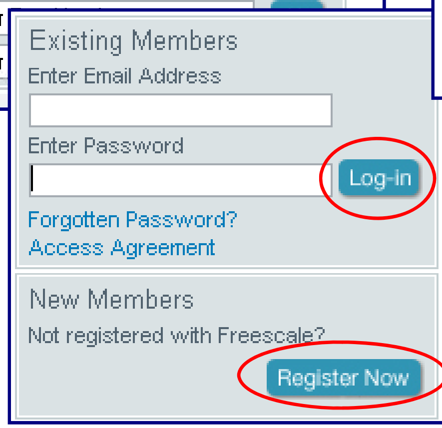
*Energy and signals travel in the **spaces** not the traces"*

Ralph Morrison

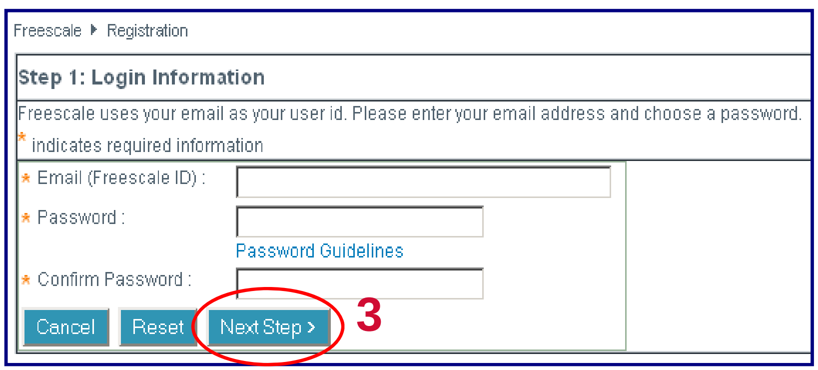
At **www.freescale.com**

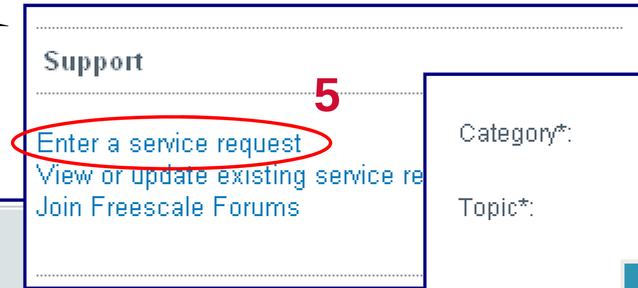
Most of your problems solved in less than **24 hours!**

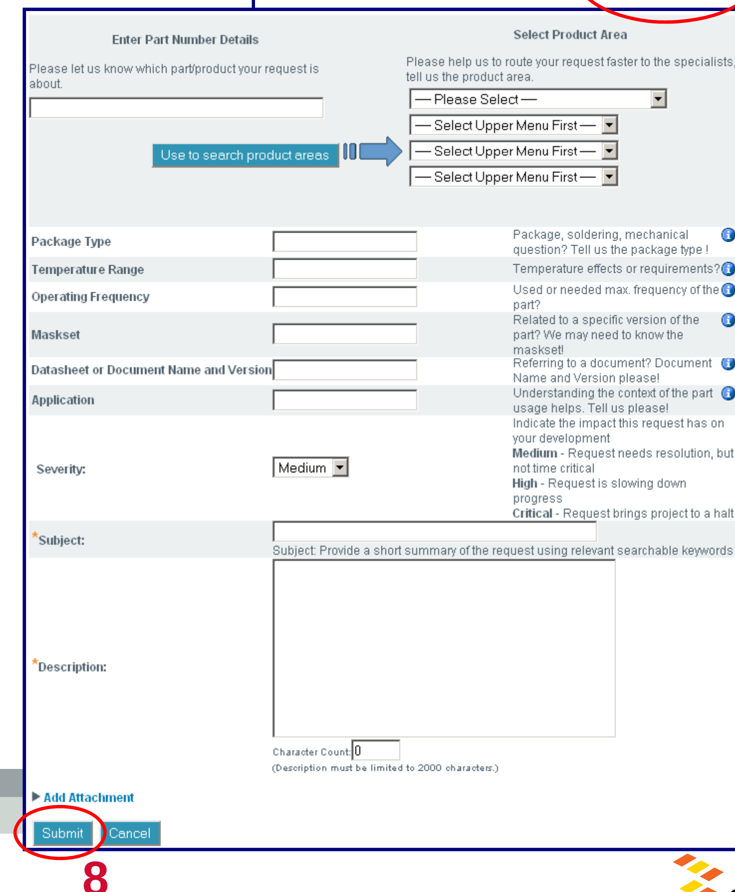




Use the **Freescale Web Technical Support!**







7 Enter your request details here!

Email: support@freescale.com