Effective Printed Circuit Board Design: Techniques to Improve Performance
AMF-ENT-T0040

Daniel Beeker
Senior Field Applications Engineer
Agenda

- Changes on the Wind
- Foundation of Electronics
- Electromagnetic Field Behavior
- What’s in the Waves
- Component Placement
- Power Distribution
- Designing Good Transmission Lines
- Test Results
- New Rules of Thumb
- Reference Information
So...What the heck is this all about?
What are the biggest differentiators between your product and your competitor’s product?
Effective PCB Design: Techniques to Improve Performance

• The packaging?

• The mix of parts you choose?

• The microcontroller?
Effective PCB Design: Techniques to Improve Performance

- Software
  - Correct
  - Efficient

- PCB design!
The Printed Circuit Board?

• Usually the largest, most expensive component

• Usually the only true custom hardware component

• Usually the last piece of the hardware puzzle
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
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 completely falls 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!
Effective PCB Design: Techniques to Improve Performance

- 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
I Had an Epiphany at PCB West

• System designers do need to care about PCB design.

• Electromagnetic field behavior is not “black magic.”

• Geometry is critical.

• There are solutions that work.
About Me: Daniel Beeker

• 30+ years experience at Motorola/Freescale designing and working with microprocessor and microcontroller development systems

• 20+ years working with automotive customers in one of the most demanding embedded control environments

• Championing the cause for increased awareness of advanced design technologies

• Used to believe in *black magic*, but Ralph Morrison set me straight!

• Firmly entrenched in physics-based design philosophy
Electromagnetic Fields: The Foundation of Electronics
Effective PCB Design: Myths We Depended On

- Fields are invisible
- Fields are well behaved
- Fields follow the trace
- Fields avoid open spaces
- Fields are someone else’s problem
- Fields are only important in RF and power supply designs
- Fields are only for farmers
Is it *volts* and *amperes* …

or *electric* and *magnetic* fields?

(Slide compliments of Ralph Morrison, Consultant)
Effective PCB Design: Techniques to Improve Performance

• Fields are basic to all circuit operation

• Volts and amperes make things practical
  - We easily can measure volts and amperes
  - More difficult to measure “E” and “H” fields

• In high clock rate (and rise time) circuits, once the "quasi static" approximation does not hold true anymore, field control plays a critical role

• This must be a carefully considered part of any design

(Slide compliments of Ralph Morrison, Consultant)
Effective PCB Design: Maxwell’s Equations

\[
\oint E \cdot dA = \frac{q_{enc}}{\varepsilon_0}
\]

\[
\oint B \cdot dA = 0
\]

\[
\oint E \cdot ds = -\frac{d\Phi_B}{dt}
\]

\[
\oint B \cdot ds = \mu_0 \varepsilon_0 \frac{d\Phi_E}{dt} + \mu_0 i_{enc}
\]

(Slide compliments of http://www.physics.udel.edu/~watson/phys208/ending2.html)
Effective PCB Design: Techniques to Improve Performance

Maxwell was smart!
A Loose Field is *Not* a Friendly Field

- **unHappy field in a sphere with an opening**
  - Sneaks out and has a party

- **unHappy field in a poor coaxial cable**
  - Runs back along the outside of the cable and causes trouble

- **unHappy field in a widely spaced transmission line pair**
  - Reaches out as far as it can, looking for other paths to follow

- **unHappy field between two widely spaced PCB planes**
  - Low field density, has very little energy
Fields are Friendly?

Field is not contained and looks for trouble

(Slide compliments of Ralph Morrison, Consultant)
Effective PCB Design: Energy Management

- Fields store energy in *space*!
- Energy is *not* stored in or on the conductors
- Remember, Voltage is a measure of the field density per unit volume

A capacitor is:
A conductor geometry that concentrates the storage of electric field energy

In a capacitor
Field energy is stored in the space between the plates

An inductor is:
A conductor geometry that concentrates the storage of magnetic field energy

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 D field is a displacement current; creating 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)
effective PCB Design: Techniques to Improve Performance

• Why does field 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)
Effective PCB Design: Techniques to Improve Performance

- 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)
Effective PCB Design: Transmission Line Properties

- 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)
Effective PCB Design: Techniques to Improve Performance

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)
Effective PCB Design: Techniques to Improve Performance

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 (signal integrity) problems
- This is called energy management

Local sources of energy:
- Decoupling capacitors
- There is also energy available from ground/power plane capacitance

New problem:
- It takes time to move this energy from storage to a load

(Slide compliments of Ralph Morrison, Consultant)
Effective PCB Design: Techniques to Improve Performance

• How long does it take?

• Wave velocity
  - For traces on a circuit board \( v = c / \varepsilon^{1/2} \)
  - Where \( c \) is the velocity of light and \( \varepsilon \) is the relative dielectric constant

• \( v = 150 \text{ mm/ns} \) or \( 6”/\text{ns} \)

• 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)
Effective PCB Design:
Techniques to Improve Performance

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 takes 30 round trips on that line to bring current level up to near one amp
  - That is 600 ps, assuming zero rise time

(Slide compliments of Ralph Morrison, Consultant)
• Note: This is not a curve, but a series of step functions. The amplitude of the step is determined by the impedance of the transmission line and the width of the step is determined by the length of the transmission line and a two way transition for the wave.

(Slide compliments of Ralph Morrison, Consultant)
Effective PCB Design: Techniques to Improve Performance

• 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)
Electromagnetic Fields: Transmission lines

- 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)
Electromagnetic Fields: Energy Management

• 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
  - Space between the wirebonds
  - Between layers of Substrate (BGA) or leadframe (QFP)
  - Power planes if present
  - Local bypass capacitors
  - Field energy stored across the PCB structure
  - Bulk storage capacitors
  - Finally the power supply

• We have to keep the field happy and contained as far up the food chain as we can, to reduce system noise.

(Slide compliments of Ralph Morrison, Consultant)
Effective PCB Design: Techniques to Improve Performance

Antenna Size vs. Frequency

<table>
<thead>
<tr>
<th>Frequency</th>
<th>¼ wave length</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 Hertz</td>
<td>246,000,000 feet (46,591 miles) Almost 6 times around the earth</td>
</tr>
<tr>
<td>10 Hertz</td>
<td>24,600,000 feet (4,659 miles) Almost from Detroit to Honolulu</td>
</tr>
<tr>
<td>100 Hertz</td>
<td>2,460,000 feet (466 miles) Almost from Detroit to New York</td>
</tr>
<tr>
<td>1 KHz</td>
<td>246,000 feet (46.6 miles) Almost from Novi to Flint</td>
</tr>
<tr>
<td>10 KHz</td>
<td>24,600 feet (46.6 miles) Almost from Freescale Novi to Walled Lake</td>
</tr>
<tr>
<td>100 KHz</td>
<td>2,460 feet (0.466 miles) Almost from the Freescale Novi to Meadowbrook Road</td>
</tr>
<tr>
<td>1 MHz</td>
<td>246 feet (0.0466 miles) Less than a football field</td>
</tr>
<tr>
<td>10 MHz</td>
<td>24.6 feet Across the room</td>
</tr>
<tr>
<td>100 MHz (TTL Logic)</td>
<td>2.46 feet Less than a yard</td>
</tr>
<tr>
<td>1 GHz (BiCMOS Logic)</td>
<td>0.246 feet (2.952 inches) Less than your finger</td>
</tr>
<tr>
<td>10 GHz (GaAs Logic)</td>
<td>0.0246 feet (0.2952 inches) Less than the diameter of a pencil</td>
</tr>
<tr>
<td>100 GHz (nanometer geometry HCMOS)</td>
<td>0.00246 feet (0.0295 inches) Half the thickness of a standard FR4 PCB</td>
</tr>
</tbody>
</table>
## Effective PCB Design: Techniques to Improve Performance

### Antenna Size vs. Frequency

<table>
<thead>
<tr>
<th>Frequency</th>
<th>¼ wave length</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 MHz</td>
<td>HMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 100 nanoseconds</td>
<td>24.6 feet</td>
</tr>
<tr>
<td>rise time distance, 100 feet</td>
<td>24.6 feet</td>
</tr>
<tr>
<td>100 MHz (TTL Logic)</td>
<td>UDR HCMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 10 nanoseconds</td>
<td>2.46 feet</td>
</tr>
<tr>
<td>rise time distance, 10 feet</td>
<td>2.46 feet</td>
</tr>
<tr>
<td>1 GHz (BiCMOS Logic)</td>
<td>IDR HCMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 1 nanosecond</td>
<td>0.246 feet (2.952 inches)</td>
</tr>
<tr>
<td>rise time distance, 1 foot</td>
<td>0.246 feet (2.952 inches)</td>
</tr>
<tr>
<td>10 GHz (GaAs Logic)</td>
<td>65 nM HCMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 100 picoseconds</td>
<td>0.0246 feet (0.2952 inches)</td>
</tr>
<tr>
<td>rise time distance, 1.2 inches</td>
<td>0.0246 feet (0.2952 inches)</td>
</tr>
<tr>
<td>100 GHz</td>
<td>32 nM HCMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 10 picoseconds</td>
<td>0.00246 feet (0.0295 inches)</td>
</tr>
<tr>
<td>rise time distance, 0.12 inches</td>
<td>0.00246 feet (0.0295 inches)</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>
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, every structure in the system can be an good antenna.

Since TTL days:

- Four order magnitude change in switching speeds
- Almost no changes in PCB or system design philosophy
Wave Reflection

Reflection Coefficient

Reflection: \[ \rho = \frac{Z_2 - Z_1}{Z_2 + Z_1} \]

- When \( Z_2 = Z_1 \) then \( \rho = 0 \) or *no reflection*
- When \( Z_2 > Z_1 \) then \( \rho > 0 \) or *a plus reflection*
- When \( Z_2 < Z_1 \) then \( \rho < 0 \) or *a negative reflection*

(Slide compliments of Ralph Morrison, Consultant)
Effective PCB Design:
Wave on a Transmission Line

Low impedance source to high impedance load
Wave reflects and voltage doubles

(Slide compliments of Ralph Morrison, Consultant)
• From a low impedance to a high impedance, the wave voltage is doubled and reflected. From a high impedance to a low impedance, the wave voltage is inverted and reflected.

• This is called “ringing” and continues until all of the energy is either transferred to the receiver, converted to heat in the dielectric, or radiates.

(Slide compliments of Ralph Morrison, Consultant)
Electromagnetic Fields: Rise Time Distance

• Now, what does this really mean?
• Rise time distance is how far the wave travels by the time it reaches full amplitude.
  – Determined by the *switching speed* of the output driver
• Let’s look at this from a switching speed vs. lumped distance perspective.
• Remember, lumped distances are basically the size of a discontinuity which remains invisible to the energy flow.
• To prevent problems on uncontrolled impedance transmission lines, the load must be less than 1/12 of the wavelength
  – The voltage developed is less than ½ of the output voltage, so the reflection is less than the output voltage
  – One reflection and the transmission line is stable
• How far is that for a given switching speed?
### Effective PCB Design:
Techniques to Improve Performance

Logic Families / Rise Time / Max Length

<table>
<thead>
<tr>
<th>DEVICE TYPE</th>
<th>RISETIME</th>
<th>Max Line Length Inner (inch/mm)</th>
<th>Max Line Length Outer (inch/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard TTL</td>
<td>5.0 nSec</td>
<td>7.27 / 185</td>
<td>9.23 / 235</td>
</tr>
<tr>
<td>Schottky TTL</td>
<td>3.0 nSec</td>
<td>4.36 / 111</td>
<td>5.54 / 141</td>
</tr>
<tr>
<td>10K ECL</td>
<td>2.5 nSec</td>
<td>3.63 / 92</td>
<td>4.62 / 117</td>
</tr>
<tr>
<td>ASTTL</td>
<td>1.9 nSec</td>
<td>2.76 / 70</td>
<td>3.51 / 89</td>
</tr>
<tr>
<td>FTTL</td>
<td>1.2 nSec</td>
<td>1.75 / 44</td>
<td>2.22 / 56</td>
</tr>
<tr>
<td>BICMOS</td>
<td>0.7 nSec</td>
<td>1.02 / 26</td>
<td>1.29 / 33</td>
</tr>
<tr>
<td>10KH ECL</td>
<td>0.7 nSec</td>
<td>1.02 / 26</td>
<td>1.29 / 33</td>
</tr>
<tr>
<td>100K ECL</td>
<td>0.5 nSec</td>
<td>.730 / 18</td>
<td>.923 / 23</td>
</tr>
<tr>
<td>GaAs</td>
<td>0.3 nSec</td>
<td>.440 / 11</td>
<td>.554 / 14</td>
</tr>
</tbody>
</table>

(Calculated assuming a nominal Er = 4.1)

(Slide compliments of Rick Hartley, Consultant)
Effective PCB Design: Techniques to Improve Performance

Switching Frequency vs. Lumped Distance

<table>
<thead>
<tr>
<th>Frequency</th>
<th>1/12 wave length</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 MHz</td>
<td>8.2 feet</td>
</tr>
<tr>
<td>HMOS</td>
<td>In the room</td>
</tr>
<tr>
<td>Rise time equivalent, 100 nanoseconds</td>
<td></td>
</tr>
<tr>
<td>Rise time distance, 100 feet</td>
<td></td>
</tr>
<tr>
<td>100 MHz (TTL Logic)</td>
<td>0.82 feet</td>
</tr>
<tr>
<td>UDR HCMOS</td>
<td>Anywhere on the board</td>
</tr>
<tr>
<td>Rise time equivalent, 10 nanoseconds</td>
<td></td>
</tr>
<tr>
<td>Rise time distance, 10 feet</td>
<td></td>
</tr>
<tr>
<td>1 GHz (BiCMOS Logic)</td>
<td>0.082 feet (0.984 inches)</td>
</tr>
<tr>
<td>IDR HCMOS</td>
<td>Pretty close</td>
</tr>
<tr>
<td>Rise time equivalent, 1 nanosecond</td>
<td></td>
</tr>
<tr>
<td>Rise time distance, 1 foot</td>
<td></td>
</tr>
<tr>
<td>10 GHz (GaAs Logic)</td>
<td>0.0082 feet (0.0984 inches)</td>
</tr>
<tr>
<td>65 nm HCMOS</td>
<td>In the package</td>
</tr>
<tr>
<td>Rise time equivalent, 100 picoseconds</td>
<td></td>
</tr>
<tr>
<td>Rise time distance, 1.2 inches</td>
<td></td>
</tr>
<tr>
<td>100 GHz</td>
<td>0.00082 feet (0.00984 in. or 250 µm)</td>
</tr>
<tr>
<td>32 nm HCMOS</td>
<td>On the die</td>
</tr>
<tr>
<td>Rise time equivalent, 10 picoseconds</td>
<td></td>
</tr>
<tr>
<td>Rise time distance, 0.12 inches</td>
<td></td>
</tr>
</tbody>
</table>
Effective PCB Design: Techniques to Improve Performance

- For energy to be delivered from a storage device:
  - The wave requesting the energy (a dip in the power supply caused by the switching event) has to travel to the source and back to the switch.

- It’s a two-way trip!
Effective PCB Design: Techniques to Improve Performance

Switching Frequency vs. Power Source

<table>
<thead>
<tr>
<th>Frequency</th>
<th>1/20 wave length</th>
</tr>
</thead>
<tbody>
<tr>
<td>10 MHz</td>
<td>HMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 100 nanoseconds</td>
<td>6.15 feet</td>
</tr>
<tr>
<td>Rise time distance, 100 feet</td>
<td>Somewhere in the room</td>
</tr>
<tr>
<td>100 MHz (TTL Logic)</td>
<td>UDR HCMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 10 nanoseconds</td>
<td>.615 feet</td>
</tr>
<tr>
<td>Rise time distance, 10 feet</td>
<td>Somewhere on the board, should be routed as co-planar pairs</td>
</tr>
<tr>
<td>1 GHz (BiCMOS Logic)</td>
<td>IDR HCMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 1 nanosecond</td>
<td>0.0615 feet (0.738 inches)</td>
</tr>
<tr>
<td>Rise time distance, 1 foot</td>
<td>Width of your finger, time to look at small geometry capacitors and power islands</td>
</tr>
<tr>
<td>10 GHz (GaAs Logic)</td>
<td>65 nm HCMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 100 picoseconds</td>
<td>0.00615 feet (0.0738 in. or 1874.5 µm)</td>
</tr>
<tr>
<td>Rise time distance, 1.2 inches</td>
<td>In the package</td>
</tr>
<tr>
<td>100 GHz</td>
<td>32 nm HCMOS</td>
</tr>
<tr>
<td>Rise time equivalent, 10 picoseconds</td>
<td>0.000615 feet (0.00738 in. or 187.45 µm)</td>
</tr>
<tr>
<td>Rise time distance, 0.12 inches</td>
<td>On the die</td>
</tr>
</tbody>
</table>
Effective PCB Design: Techniques to Improve Performance

- If the energy source is not inside the 1/20 wavelength distance, there will be radiated energy caused by the switching event.

- The job of the PCB designer is to minimize the amount of energy by managing the power delivery system for each type of switching event.

- As the geometry of the ICs we use continues to shrink, so does the area of effective power delivery.

- Well-defined power delivery transmission lines and small geometry, low impedance field storage devices are essential.

- Even if they are outside of the “zone,” they can minimize the amount of radiated energy.
Fields are Friendly!

A contained field is a friendly field:
- Happy field in a sphere
- Happy field in a good coaxial cable
- Happy field in a closely spaced transmission line pair
- Happy field between two closely spaced PCB planes
Fields are Friendly!

An equipotential surface around a charged sphere

Quiet at home, no reason to roam

(Slide compliments of Ralph Morrison, Consultant)
Fields are Friendly!

Coaxial Transmission

No radiation

E and H fields are contained

Current return path must be on sheath.

Quiet at home, no reason to roam

(Slide compliments of Ralph Morrison, Consultant)
Fields are Friendly!

Fields concentrate under the traces and there is little crosstalk.

Fields do not penetrate the plane.

Quiet at home, no reason to roam

(Slide compliments of Ralph Morrison, Consultant)
Fields are Friendly?

Fields need to be carefully managed:

- Every connection must be treated as part of a transmission line pair
- Field volumes (read *transmission line impedance*) must be carefully managed
- Each discontinuity (read *change in transmission line GEOMETRY*) results in reflections
- Each segment of this geometry must have enough field energy delivered to match the field density (read *voltage*) from the driver

- This all takes TIME

Yes, this is now a **four-dimensional geometric** design problem
Electromagnetic Fields: Energy and Logic Signals

• The transmission of a logic signal means that field energy is sent out on a transmission line
  - Logic drivers should be treated the same as any power source

• This is true even if the line is un-terminated
  - The driver does not know what is at the end of the transmission line
  - The driver only sees a short circuit until after a reflection occurs

• This energy must be transmitted to the receiver or lost in heat or radiation – it cannot be returned to the driver
Well-Defined Transmission Lines

• Signal traces *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

• Unless a transmission line is required to be controlled impedance (read *receiver is more than 1/12th wavelength away*), the goal should be the lowest possible (practical) impedance.

This is a very serious problem and a big change from normal board design philosophy.
Electromagnetic Fields: Transmission lines

• Good news is that any discontinuity that is less than 1/12th wavelength is virtually invisible to the signal
  - Routing schemes need to be driven by the transistor geometry

• Any failure to insure that both signal and ground copper are contiguous results in large discontinuities that will cause signal integrity and EMC issues
  - Vertical transitions can not be not excluded
    ▪ This is the most common mistake made in otherwise good designs
Electromagnetic Fields:
How to Use this Wonderful Information
Effective PCB Design: Techniques to Improve Performance

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
Effective PCB Design:
Techniques to Improve Performance

- 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 that 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.
Effective PCB Design: Techniques to Improve Performance

- 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
Effective PCB Design: Techniques to Improve Performance

- Schematic must be evaluated during layout
- Pin assignment to connector signals

Ideal connector pin assignment:

- \text{PGA}SGSGSGS\text{GP}
- GSGSGSGSGG

Not exactly economical or practical

More practical and fewer ground pins:

- SSSGSSGSS\text{P}
- SGSSSGSSSG\text{P}

Each signal is only 1 pin spacing from ground
Effective PCB Design: Techniques to Improve Performance

- 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

This can be applied when you are not allowed sufficient returns, but will improve EMC
Effective PCB Design: Techniques to Improve Performance

- 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
Effective PCB Design: Techniques to Improve Performance

- 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
Effective PCB Design: Techniques to Improve Performance

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
  - Use the largest value capacitor in the smallest package allowed by manufacturing and reliability
- 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
Effective PCB Design: Techniques to Improve Performance

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!

Let’s move on to actually routing the board…
Effective PCB Design: Techniques to Improve Performance

- The first and most important job 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
Effective PCB Design: Techniques to Improve Performance

PCB signal transmission line routing

Input Connector

Minimize loop area, or field volume
Effective PCB Design: Techniques to Improve Performance

- 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
Effective PCB Design: Techniques to Improve Performance

PCB signal transmission line routing

Input Connector

Input Filter

VDD

GND
Effective PCB Design: Techniques to Improve Performance

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
Effective PCB Design: Techniques to Improve Performance

PCB signal transmission line routing

Input Connector

Input Filters

Routed Triplet

No crosstalk here!
**Effective PCB Design:**
Techniques to Improve Performance

- 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!

Note: All field lines actually terminate at 90 degree angles
You really want to make sure that the field energy is coupling to the conductor you choose!

Note: All field lines actually terminate at 90 degree angles

Maybe a “triplet” makes sense?
Effective PCB Design: Techniques to Improve Performance

PCB signal transmission line routing

Input Connector

Secondary Side Input Filters

Routed Triplet

No crosstalk here! Well-defined transmission lines
Effective PCB Design: Techniques to Improve Performance

PCB signal transmission line routing

Input Connector

Input Filters

Routed Triplet

No crosstalk here!

Referenced to second ground pin
Effective PCB Design: Techniques to Improve Performance

PCB signal transmission line routing

Routed Triplets

GND

VDD

Local Charge Source

MCU

Mini-Plane

Routed Triplets

VDDI

GND

Internal Regulator

Local Charge Source
Effective PCB Design: Techniques to Improve Performance

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 two via to adjacent layer ground
- Minimize the volume of the signal transmission network
Effective PCB Design: Techniques to Improve Performance

- PCB signal transmission line routing
  - Oscillator ground must be routed to MCU pin, not tied to System Ground
  - Connect System Ground on the opposite side of the MCU pin

Grey traces = primary side
Red traces = secondary side (or adjacent layer)
Effective PCB Design: Techniques to Improve Performance

PCB signal transmission line routing

- Oscillator ground must be routed to MCU pin, not tied to System Ground
- Connect System Ground on the opposite side of the MCU pin

Grey traces = primary side
Why this works

- Crystal output from MCU is the POWER source
  - Energy flows out, between the MCU output signal trace and MCU oscillator ground trace to the crystal or resonator
- Crystal becomes POWER source
  - Energy flows back, between the crystal output signal trace and the oscillator ground trace to the MCU input pin
- This is a closed loop system, and SYSTEM ground is not required
  - Lower impedance for the connecting transmission lines
  - Smaller loop area
  - Significantly improved robustness
- It is always about “Where does the energy come from?”
- This rule should be applied to the entire design
  - Ground for any signal is determined by the power source return
Sure, But Does This Stuff REALLY Work?
Testing and Evaluation
Effective PCB Design: 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
Effective PCB Design: Techniques to Improve Performance

KIT33812ECUEVME Reference Design
Effective PCB Design: Techniques to Improve Performance

KIT33812ECUEVME Reference Design
Primary Silk
Effective PCB Design: Techniques to Improve Performance

KIT3382ECUEVME Reference Design
Primary Side
Effective PCB Design: Techniques to Improve Performance

KIT33812ECUEVME Reference Design
Secondary Side
Effective PCB Design: Techniques to Improve Performance

KIT33812ECUEVME Reference Design

Professional Testing
10 Meter Radiated Emissions
30-1000MHz Class A Horizontal Plot

Operator: Larry Fuller
04:13:55 PM, Monday, December 14, 2009
Effective PCB Design: Techniques to Improve Performance

KIT33812ECUEVME Reference Design

Professional Testing
3 Meter Radiated Emissions
1-15GHz Class A Horizontal Plot

Company - Technology International
Model # - KIT33812 ECUEVME
Description -
Project # - 10644-10
Voltage - 230 VAc 50 Hz

Operator: Larry Fuller
04:28:11 PM, Monday, December 14, 2009
Effective PCB 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
Effective PCB Design: Techniques to Improve Performance

EMC Test Board Schematic
Effective PCB Design: Techniques to Improve Performance

EMC Test Board Layout
Effective PCB Design: Techniques to Improve Performance

Two-layer EMC Test Board Radiated Emissions
Effective PCB Design: Techniques to Improve Performance

Two-layer EMC Test Board Conducted Emissions

Conducted Emissions (CE)

Frequency (MHz)

Two-layer EMC Test Board Conducted Emissions
Effective PCB Design:
Techniques to Improve Performance

- **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
Effective PCB Design: Techniques to Improve Performance

EMC Test Board Layout
Effective PCB Design:
Techniques to Improve Performance
Effective PCB Design: Techniques to Improve Performance

Conducted Emissions (CE)

2- vs. 4-layer EMC Test Board Conducted Emissions
WOW!

What would you do for 30 db?
Effective PCB Design: 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 LEDs 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 e-mail.
- This is a four-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

<table>
<thead>
<tr>
<th>Frequency</th>
<th>¼ wave length</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 Hertz</td>
<td>246,000,000 feet (46,591 miles)</td>
</tr>
<tr>
<td>10 Hertz</td>
<td>24,600,000 feet (4,659 miles)</td>
</tr>
<tr>
<td>100 Hertz</td>
<td>2,460,000 feet (466 miles)</td>
</tr>
<tr>
<td>1 KHz</td>
<td>246,000 feet (46.6 miles)</td>
</tr>
<tr>
<td>10 KHz</td>
<td>24,600 feet (4.659 miles)</td>
</tr>
<tr>
<td>100 KHz</td>
<td>2,460 feet (0.466 miles)</td>
</tr>
<tr>
<td>1 MHz</td>
<td>246 feet (0.0466 miles)</td>
</tr>
<tr>
<td>10 MHz</td>
<td>24.6 feet</td>
</tr>
<tr>
<td>100 MHz</td>
<td>2.46 feet</td>
</tr>
<tr>
<td>1 GHz</td>
<td>0.246 feet (2.952 inches)</td>
</tr>
<tr>
<td>10 GHz</td>
<td>0.0246 feet (0.2952 inches)</td>
</tr>
<tr>
<td>100 GHz</td>
<td>0.00246 feet (0.0295 inches)</td>
</tr>
</tbody>
</table>
Effective PCB Design: Techniques to Improve Performance

EMC Test Results

Professional Testing
10 Meter Radiated Emissions
30-1000MHz Class A Vertical Plot

Company: Technology International
Model #: LFBDMPGMR
Description: 
Project #: 09470-10
Voltage: 120 VAc 60 Hz

Amplitude (dBuV/m)

Operator: Larry Fuller
03:51:32 PM, Monday, March 16, 2009

Frequency (Hz)
New Mods.
Effective PCB Design:
Techniques to Improve Performance
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 %

<table>
<thead>
<tr>
<th>EUT Face</th>
<th>Frequency Range</th>
<th>80-200 MHz</th>
<th>200-1000 MHz</th>
<th>1.4-2.0 GHz</th>
<th>2.0-2.7 GHz</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>3 V/m</td>
<td>3 V/m</td>
<td>V/m</td>
<td>V/m</td>
<td>V/m</td>
</tr>
<tr>
<td>Illuminated</td>
<td>Horizontal</td>
<td>Vertical</td>
<td>Horizontal</td>
<td>Vertical</td>
<td>Horizontal</td>
</tr>
<tr>
<td>Front</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>Right</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>Rear</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>Left</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
</tbody>
</table>

Test Results: Pass X Fail

The EUT met performance criteria:
- Criteria A
- 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"
Effective PCB Design:
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
Effective PCB Design: More PC Board Considerations

- Use minimum trace widths and spacing for signal transmission lines
  - Refer to PCB fabricator’s capabilities without a cost adder
  - Same thing goes for drill sizes and pad rings
  - May be defined by either customer or internal requirements
  - Wider traces for power supply transmission line pairs
  - Provides maximum trace density

- Make room for all of those ground traces!
Most PC boards are “foil laminated”

(Slide compliments of Rick Hartley, Consultant)
Effective PCB Design: More PC Board Considerations

- Four-layer boards:
  - Made from a two-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: 
More PC Board Considerations

• Four-layer boards:
  - Most effective stackup has one ground layer
  - L2 ground means that L1 and L3 are one dielectric from ground
  - L4 must be routed as a single layer board, with following ground traces (triplets?)
  - Ground transition vias are required when signals go from layer 4 to any other layer to insure the transmission line is continuous
Effective PCB Design: More PC Board Considerations

- **Six-layer boards:**
  - Most effective stackup has two ground layers
  - L2 as ground means that L1 and L3 are one dielectric from ground
  - L5 as ground means that L4 and L6 are one dielectric from ground
  - Ground transition vias are required when signals go from one ground reference group (L1-L2-L3 or L4-L5-L6) to any other layer in the other group) to insure the transmission line is continuous
Effective PCB Design: More PC Board Considerations

- Layer count determinations
- Technology of the devices used
- Trace density
- EMC certification level
  - Consumer/commercial
  - Automotive
  - Aviation
  - Military

All must be considered, not just trace density!
Effective PCB Design: 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 \textit{not} reduced by reducing IC geometries!
I repeat:

System cost is **not** reduced by reducing IC geometries!
Effective PCB Design: More PC Board Considerations

- Using planes
  - Both power and ground can be used as signal references
  - Transition from one reference plane to another requires close proximity to a bypass capacitor
    - That is the only way the energy can go!

  - Only if they are well coupled to each other
    - Capacitors
      - Impedance is less than 0.1 ohm
    - Adjacent to each other
      - Less than 10 mils
When routing signals with returns between power and ground planes, return energy will transfer as follows:

- **Tightly coupled planes**
  - Signal
  - Return
  - Ground
  - Power

- **Loosely coupled planes with cap**
  - Signal
  - Return
  - Ground
  - Power

(Slide compliments of Rick Hartley, Consultant)
Effective PCB Design: Techniques to Improve Performance

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

- When moving signals *between two different planes*, use a transfer via very near the signal via.

(Slide compliments of Rick Hartley, Consultant)
Effective PCB Design: More PC Board Considerations

- Remember, field energy moves in the space between or around the conductors and cannot go through them. 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!

Statement compliments of Ralph Morrison, Consultant
Which of these is best?

90 Degree Corner, Cut at 45 Degrees.
45 Degree Corner.
90 Degree Corner.
Radius at Corner.

(Slide compliments of Rick Hartley, Consultant)
Effective PCB Design: More PC Board Considerations

- These small discontinuities are virtually invisible for all applications in the foreseeable future.
- The best one is the 45 degree because it is easier to manufacture and easier to draw.
- The 90-degree choices tend to be victims of under or over etching, and can form failure points in the future, as well as impacting PCB yields.
- The radius is good, but most CAD packages do not support this option.
Effective PCB Design:
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!
Signal Return Path

Two-layer Microwave Style PC Board

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?
Effective PCB Design: 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
Effective PCB Design: More PC Board Considerations

- Routing over split planes, different potential
- Have to bridge with a capacitor
Effective PCB Design: Routing Differential Signals

Myth: They are coupled to each other
Differential signals are referenced to each other only when twisted pair wiring is used, NOT on PCBs

- Fact: They are coupled to ground
  - Each driver power source is coupled to ground, hence the outputs must be 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
  - Not possible on PCBs, since there is no way to subject both signals to the same interference by twisting)
Effective PCB Design: 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
  - Fact: What matters is the set up and hold time required by the devices
    - This is usually specified in time (ps)
    - Remember this? $v = 150 \text{ mm} / \text{ ns}$ or $6'' / \text{ ns}$

- For a typical 500 MHz DDR memory interface, the data lines only need to be within 500 mils of each other in length
  - Way easier than we have been led to believe

---

2 Slide compliments of Rick Hartley, Consultant
Return Path equally important in IC Package.
F1120 had 5X greater noise level than FF148

Xilinx Virtex-4 FF148
Returns spread evenly

Altera Stratix II F1120
Many regions devoid of returns

Source: BGA Crosstalk - Dr. Howard Johnson
Signal Return Path BGA Design

Rainbow 324 BGA

| A | B | C | D | E | F | G | H | I | J | K | L | M | N | P | Q | R | S | T | U | V | W | X | Y | Z |
|   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |   |

Only 14 **VSS (blue) pins**, but package met original marketing price target. Significant issues with signal integrity and EMC. Notice, only one VSS for then entire DDR interface.
Rainbow 416 BGA

Began as 324 pins ...is now 416 pins! Worked with customer to discuss need for larger package with more pins.

Used a mixture of 3x3 grids with center pin **VSS (green)** for outer balls and checkerboard approach to center power balls.
Closing Remarks and Reference Materials

PCB Design is *Not* a Black Art!
Fundamentals to Remember

- Electromagnetic fields travel in the space between the conductors, not in the conductors.
- The switching speed of the transistors determines the frequency of operation, not the clock rate.
- Signal and power connections need to be one dielectric from ground for their entire length (including layer transitions).
  - Adjacent plane
  - Co-planar trace
- In a 4-layer design, if one layer is a ground plane, the two adjacent layers are one dielectric from ground.
- The 4th layer must be routed as if it were a single layer board.
- There is no such thing as a noisy ground, just poor transmission line design.
- To quote Dr. Todd Hubing, “Thou shalt not split ground.”
- Any compromises to these rules will increase system noise and must be done as carefully considered engineering decisions.
Special Thanks to My Mentors

• **Rick Hartley** (PCB designer extraordinaire) started me down this trail in 2004 at PCB West

• **Ralph Morrison** (author, inventor and musician) 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 has provided solid evidence that Maxwell and Ralph have got it right.

• Finally, my team at Freescale. We really have come a long way!
High Speed Design Reading List


(Slide compliments of Rick Hartley, Consultant)
EMI Reading List


(Slide compliments of Rick Hartley, Consultant)
**Additional References**

- Ralph Morrison’s New Book: *Digital Circuit Boards: Mach 1 GHz*. Available from Wiley and Amazon
- The Best PCB design conference website: [http://pcbwest.com/](http://pcbwest.com/)
- Doug Smith’s website: [http://www.emcesd.com/](http://www.emcesd.com/) (He is the best at finding what is wrong! Lots of useful app notes.)
- Clemson’s Automotive Electronics website: [http://www.cvel.clemson.edu/auto](http://www.cvel.clemson.edu/auto)
- Clemson’s EMC website: [http://www.cvel.clemson.edu/emc](http://www.cvel.clemson.edu/emc)
- Missouri University of Science and Technology website: [http://www.mst.edu/about/](http://www.mst.edu/about/)
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
Summary and Q&A

• 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!

• Q&A