November, 2010

Effective PCB Design:
Techniques to Improve Performance

Daniel Beeker
Senior Field Applications Engineer
So...????

What the heck is this all about?
What are the biggest differentiators between your product and your competitor’s?
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
Largest, most expensive component

Usually the last piece of the hardware puzzle
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!
Effective PCB Design: Techniques to improve performance

► 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
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 of experience at Motorola/Freescale designing and working with Microprocessor and Microcontroller development systems

- More than 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
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
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
Maxwell’s Equations

\[ \oint E \cdot dA = \frac{q_{\text{enc}}}{\varepsilon_0} \]
\[ \oint B \cdot dA = 0 \]
\[ \oint E \cdot ds = -\frac{d\Phi_B}{dt} \]
\[ \oint B \cdot ds = \mu_0 \varepsilon_0 \frac{d\Phi_E}{dt} + \mu_0 i_{\text{enc}} \]

Slide compliments of http://www.physics.udel.edu/~watson/phys208/ending2.html
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 2 widely spaced PCB planes
  • Low field density, has very little energy
Fields are friendly?

A SHIELD ENCLOSURE WITH A HOLE

If $V_1 = V \sin \omega t$
then:

$$I = C_{18} \frac{dV_1}{dt} = C_{18} V \omega \cos \omega t$$

INDUCED CURRENT FLOW

Field is not contained and looks for trouble

Slide compliments of Ralph Morrison, Consultant
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

ENERGY MANAGEMENT:
Fields store Energy in SPACE!
Energy is NOT stored in or on the conductors
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 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
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
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
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!
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 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 \( v = \frac{c}{\sqrt{\varepsilon}} \)
- WHERE \( c \) IS THE VELOCITY OF LIGHT AND \( \varepsilon \) IS THE RELATIVE DIELECTRIC CONSTANT

\[ v = 150 \text{ mm} / \text{ns} \text{ or } 6” / \text{ns} \]

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
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 $\frac{1}{2}$ 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
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

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

Slide comments are compliments of Ralph Morrison, Consultant
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>ASTTTL</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>0.730 / 18</td>
<td>0.923 / 23</td>
</tr>
<tr>
<td>GaAs</td>
<td>0.3 nSec</td>
<td>0.440 / 11</td>
<td>0.554 / 14</td>
</tr>
</tbody>
</table>

(Calculated assuming a nominal Er = 4.1)

Slide compliments of Rick Hartley, Consultant
## 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></td>
<td>Almost 6 times around the earth</td>
</tr>
<tr>
<td>10 Hertz</td>
<td>24,600,000 feet (4,659 miles)</td>
</tr>
<tr>
<td></td>
<td>Almost from New York to Honolulu</td>
</tr>
<tr>
<td>100 Hertz</td>
<td>2,460,000 feet (466 miles)</td>
</tr>
<tr>
<td></td>
<td>Almost from New York to Detroit</td>
</tr>
<tr>
<td>1 KHz</td>
<td>246,000 feet (46.6 miles)</td>
</tr>
<tr>
<td></td>
<td>Almost from Orlando to Cocoa Beach</td>
</tr>
<tr>
<td>10 KHz</td>
<td>24,600 feet (4,659 miles)</td>
</tr>
<tr>
<td></td>
<td>Almost from the J. W. Marriott to Disney’s Magic Kingdom</td>
</tr>
<tr>
<td>100 KHz</td>
<td>2,460 feet (0.466 miles)</td>
</tr>
<tr>
<td></td>
<td>Almost from the J. W. Marriott to the Central Florida Parkway</td>
</tr>
<tr>
<td>1 MHz</td>
<td>246 feet (0.0466 miles)</td>
</tr>
<tr>
<td></td>
<td>Less than a football field</td>
</tr>
<tr>
<td>10 MHz</td>
<td>24.6 feet</td>
</tr>
<tr>
<td></td>
<td>Across the room</td>
</tr>
<tr>
<td>100 MHz</td>
<td>2.46 feet</td>
</tr>
<tr>
<td></td>
<td>Less than a yard</td>
</tr>
<tr>
<td>1 GHz</td>
<td>0.246 feet (2.952 inches)</td>
</tr>
<tr>
<td></td>
<td>Less than your finger</td>
</tr>
<tr>
<td>10 GHz</td>
<td>0.0246 feet (0.2952 inches)</td>
</tr>
<tr>
<td></td>
<td>Less than the diameter of a pencil</td>
</tr>
<tr>
<td>100 GHz</td>
<td>0.00246 feet (0.0295 inches)</td>
</tr>
<tr>
<td></td>
<td>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 Across the room</td>
</tr>
<tr>
<td>Rise time distance, 100 feet</td>
<td></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 Less than a yard</td>
</tr>
<tr>
<td>Rise time distance, 10 feet</td>
<td></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) Less than your finger</td>
</tr>
<tr>
<td>Rise time distance, 1 foot</td>
<td></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) Less than the diameter of a pencil</td>
</tr>
<tr>
<td>Rise time distance, 1.2 inches</td>
<td></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) Half the thickness of a standard FR4 PCB</td>
</tr>
<tr>
<td>Rise time distance, 0.12 inches</td>
<td></td>
</tr>
</tbody>
</table>
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 now very friendly and want to visit everywhere.
Fields are friendly!

► Fields are in the spaces not in the traces\(^1\)

► Is there something missing in our schematics?
  • Shouldn’t there be something that helps us tell the field where it needs to be?

► We need a revolution in understanding and defining schematics and circuits.
  • Something to remind us to pay closer attention

\(^1\) Statement compliments of Ralph Morrison, Consultant
Component symbols must remind us to define the entire transmission line!

Field is free to roam
Open Range!

Field is kept out of trouble
Field fenced in!
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 2 closely spaced PCB planes
AN EQUIPOTENTIAL SURFACE AROUND A CHARGED SPHERE

Quite at home, no reason to roam

Slide compliments of Ralph Morrison, Consultant
COAXIAL TRANSMISSION

No radiation

E and H fields are contained

Current return path must be on sheath.

Quite 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 don’t penetrate the plane.

Quite 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* must be carefully managed
    (yes, this is now a 3 dimensional geometric design problem)

► Fields may not be friendly, but they can be *protected* and *tamed*!
ENERGY AND LOGIC SIGNALS

THE TRANSMISSION OF A LOGIC SIGNAL MEANS THAT FIELD ENERGY IS SENT OUT ON A TRANSMISSION LINE

THIS IS TRUE EVEN IF THE LINE IS UTERMINATED

THIS ENERGY MUST BE LOST IN HEAT OR RADIATION. IT CAN NOT BE RETURNED TO THE CIRCUIT

Slide compliments of Ralph Morrison, Consultant
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?
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
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
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:
    
    \[
    \text{PGSGSGSGSGGSP} \\
    \text{GSGSGSGSGSGG}
    \]
    
    - Not exactly economical or practical
  - More practical and fewer ground pins:
    
    \[
    \text{SSSSGSSSGGSSP} \\
    \text{SGSSSGSSSGG}
    \]
    
    - 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
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
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

- 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!

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!!

- **Now to move on to actually routing the board…**
Effective PCB Design: Techniques to improve performance

PCB Signal Transmision 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
Effective PCB Design: Techniques to improve performance

**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
Effective PCB Design:
Techniques to improve performance

PCB Signal TRANSMISSION LINE Routing

Input Connector

Input Filter

GND
VDD
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!
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???
Effective PCB Design: Techniques to improve performance

PCB Signal Transmission Line Routing

Input Connector

Routed Triplet

Secondary Side Input Filters

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

MCU Mini-Plane

Routed Triplets
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 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
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

KIT33812ECUEVME Reference Design
Primary Side
KIT33812ECUEVME Reference Design
Secondary Side
KIT33812ECUEVME Reference Design

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

Operator: Larry Fuller
04:28:11 PM, Monday, December 14, 2009

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

2 Layer EMC Test Board Radiated Emissions
Effective PCB Design: Techniques to improve performance

Conducted Emissions (CE)

2 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

Radiated Emissions #1

2 VS 4 Layer EMC Test Board Radiated Emissions
Effective PCB Design:
Techniques to improve performance

2 VS 4 Layer EMC Test Board Conducted Emissions
WOW!

What would you do for 30 db?
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

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

Operator: Larry Fuller
03:51:32 PM, Monday, March 16, 2009
EMC Test Results, Yeah!!

<table>
<thead>
<tr>
<th>Test Date: March 13, 2009</th>
<th>Client: Technology International</th>
</tr>
</thead>
<tbody>
<tr>
<td>Project #: 09470-10</td>
<td>Supervisor: Jason Anderson</td>
</tr>
<tr>
<td>EUT: LFBDMPGMR</td>
<td>Technician: Dan Keenan</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>EUT Power Source</th>
<th>120VAC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ambient Temperature</td>
<td>22.6 °C</td>
</tr>
<tr>
<td>Barometric Pressure</td>
<td>29.97 inches</td>
</tr>
<tr>
<td>Relative Humidity</td>
<td>55 %</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Frequency Range</th>
<th>3 V/m</th>
<th>5 V/m</th>
<th>5 V/m</th>
</tr>
</thead>
<tbody>
<tr>
<td>80-200 MHz</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>200-1000 MHz</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.4-2.0 GHz</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.0-2.7 GHz</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **EUT Face Illuminated**
  - 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 [ ]

- **Criteria A**: [X]
- **Criteria B**: [ ]
- **Criteria C**: [ ]
- **Manufacturer's 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.
Effective PCB Design: Techniques to improve performance

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

Copper Foil

25 to 45 Mil Thick
‘C’ Stage Core

‘C’ Stage Core

‘C’ Stage Core

Most PC Boards are “Foil Laminated”

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!
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 w/ 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!!!

![Signal Return Ground](image1)

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

![Signal Return Ground](image2)

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
Effective PCB Design: Techniques to improve performance

- PCB Trace Corners -

90 Degree Corner, Cut at 45 Degrees.

45 Degree Corner.

90 Degree Corner.

Radius at Corner.

Which of these is BEST?

Slide compliments of Rick Hartley, Consultant
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
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!!
Where does signal’s return current flow?

Slide compliments of Rick Hartley, Consultant
Signal Return Path

► 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? $v = 150 \text{ mm} / \text{ns} = 6\" / \text{ns}$

- For a typical 500 MHz DDR memory interface, the data lines *only* need to be within 500 mils of each other in length \(^2\)
  - 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!
Effective PCB Design:
Techniques to improve performance

► 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!
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


2 Slide compliments of Rick Hartley, Consultant
EMI Reading List


5. EMC for Product Engineers - Tim Williams (Newnes Publishing) - ISBN 0-7506-2466-3


*2 Slide compliments of Rick Hartley, Consultant*
Some additional references you may find useful:

  Ralph Morrison’s website

- [http://pcbwest.com/](http://pcbwest.com/)
  Best PCB design conference website

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

  IEEE EMC Society website

- [http://www.cvel.clemson.edu/auto](http://www.cvel.clemson.edu/auto)
  Clemson’s Automotive Electronics website

- [http://www.cvel.clemson.edu/emc](http://www.cvel.clemson.edu/emc)
  Clemson’s EMC website

- [http://www.mst.edu/about/](http://www.mst.edu/about/)
  Missouri University of Science and Technology website

- [http://www.ipc.org/default.aspx](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