Innovative Defense Techniques for Damping Digital to RF Crosstalk

Contributors

• Davy Pissoort; KU Leuven: davy.pissoort@khbo.be
• Hany Fahmy, Jan Van Hese; Agilent Technologies: hany_fahmy@agilent.com Jan_vanhese@agilent.com
• Mehdi Mechaik, Henry Zeng, Charlie Shu, Charles Jackson; NVIDIA Corporation: mmechaik@nvidia.com hzeng@nvidia.com cshu@nvidia.com cjackson@nvidia.com
Background

- Design of multiple antennas for mobile device communications on PCBs are major tasks due to the many sources of noise:
  - Skew, rise-fall time mismatch, delays
  - Reflections
  - Crosstalk
  - Delta-I noise
  - Radiation
- Tasks above can be overcome by adopting early simulations in pre-layout stage of PCB designs

Purpose

- This presentation will show:
  - An analysis of the main noise sources on a PCB
  - How to reduce the noise coupling
  - How to make the communication link more immune to EMI problems
- Reducing the EMI radiation effects induced on the PCB antennas is a major challenge due to:
  - Difficulty of containing radiation at high frequencies
  - Small space available to design necessary circuits
PCB Antenna Design Challenges

• PCB antennas need to be small in size at frequencies of interest for multiple bands:
  – BT
  – GSM, ...
• They rely heavily on ground planes:
  – Shape of ground plane affects antenna performance
  – Size of ground plane affects resonance frequencies
• This has important consequences wrt EMI:
  – ‘Antenna currents’ can spread over total ground plane
  – Unwanted noise coupling caused by overlapping antenna and return currents
• PCBs contain high speed buses

PCB Antenna Design (Contn’d)

• Coupling exists between analog and digital interfaces on PCBs
• In receive mode antenna sensitivity is further degraded by:
  – Low voltage input level (microvolts)
  – Coupling from digital interfaces
  – FCC power level limitations
• In transmit mode antenna performance is further degraded by:
  – Coupling between output high RF power and digital PCB interfaces
  – Antenna mismatches
PCB Antenna Design Solutions

- Partition PCB area by component interfaces
- Analyze the return current distribution for both RF and digital interfaces
- Place noisy digital interfaces away from analog parts to reduce coupling
- Minimize return path discontinuities in ground plane
- Minimize ground plane sharing between RF and digital interfaces

EMI Design process at NVIDIA

- Simulate design early on to predict EMI problems
- Estimate non-measurable quantities like current distribution
- Identify performance failure points
- Optimize design though multiple simulation iterations with accelerated computations using NVIDIA GPU hardware
- Measure and certify product
**PCB Antenna Geometry**

- The antenna used is a folded, multiband planar monopole antenna.
- Antenna is optimized for 900MHz, 1.7GHz-2.5GHz.
- It is 20mm x 8mm x 4mm.
- It is placed on one side of a ground plane.
- Antenna support material is a low dielectric ~ 2.2 dk.

---

**Influence of Solid Ground Plane Shape**

- Width is kept at 5 cm.
- Length is varied from 6cm to 11.5cm.
- Length is kept at 8 cm.
- Width is varied from 4.5cm to 6cm.
Influence of Antenna Offset

Influence of U-Shaped Ground Plane

What happens if we move this edge inward?

Moving Edge Inward
Influence of U- to Γ-Shaped Ground Plane

What happens if we make this leg shorter?

The 53mm and 73mm cases correspond to actual PCB designs.

Influence of Γ-Shaped Ground Plane

- Plotting current distribution shows hot spot regions where not to place digital components

Current distribution on GP at 900 MHz

Current distribution on GP at 2.4GHz
Coupling between Antenna and Other Interfaces

• Shown is a real PCB which is representative of wireless applications and allows study of coupling effects
• PCB has 4-bit DDR memory buses and other differential IO interfaces
• The two legs on both sides are ground extensions to other antennas and components

Coupling between Antenna and Other Interfaces

• Four main coupling mechanisms that could cause the crosstalk between digital signals and on-board antennas:
  – Near-field coupling “through the air” above the PCB
  – Coupling by waveguide modes between the different ground planes of the PCB
  – Coupling by the return currents of signal interfaces and currents induced in the ground-planes by antennas
  – Coupling by currents induced on the PCB by power and ground pins of active components/chips
Influence of Shielding on Antenna Coupling

• To look at near-field coupling and coupling by waveguide modes:
  – A perfect PEC shielding-can is placed above all traces on the PCB top side (shorted to ground plane)
  – A perfect PEC guard-ring inside the PCB substrate and around all traces (shorted to ground planes)

Influence of GP on Antenna Coupling

Current profiles for full GP
Influence of GP on Antenna Coupling

Coupling antenna/traces for full GP

Antenna currents for GP with slot
Influence of GP on Antenna Coupling

Coupling antenna/traces for GP with slot

±15 dB increase in coupling!

Influence of GP on Antenna Coupling

Time domain noise voltage (in V) at antenna port. Traces are excited with a pseudo-random bit sequence of 2Gbit/s
**Influence of GP on Antenna Coupling**

- Coupling between the traces’ and antenna’s current paths is the main contributing factor of noise
- Hence, during the placement stage of components on the PCB one has to carefully select the place where digital interfaces are placed
- Need to avoid any current return path discontinuities to prevent large current coupling

**Reduction of Antenna and Bus Coupling**

- To reduce coupling between antenna and digital buses, four Bytes are excited differently:
  - Group1: input ports on left side of top layer, time delay = 0
  - Group2: input ports on right side of top layer, time delay = Δ
  - Group3: input ports on left side of bottom layer, time delay = 2Δ
  - Group4: input ports on right side of bottom layer, time delay = 3Δ
- Δ can have different values for different cases
Reduction of Antenna and Bus Coupling

Coupling between traces and antenna are significantly reduced by about 50% up to 2.5 GHz by choosing appropriate values of Δ (delay) among the various groups.

Transfer function from all trace inputs to Antenna Port

Group 1: Blue, Group 2: Green, Group 3: Red, Group 4: Cyan

<table>
<thead>
<tr>
<th></th>
<th>Blue</th>
<th>Green</th>
<th>Red</th>
<th>Cyan</th>
</tr>
</thead>
<tbody>
<tr>
<td>Group1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Group2</td>
<td>0</td>
<td>235</td>
<td>470</td>
<td>705</td>
</tr>
<tr>
<td>Group3</td>
<td>0</td>
<td>470</td>
<td>940</td>
<td>1410</td>
</tr>
<tr>
<td>Group4</td>
<td>0</td>
<td>705</td>
<td>1410</td>
<td>2115</td>
</tr>
</tbody>
</table>

Conclusions

- Coupling between traces and antenna are significantly reduced by about 50% up to 2.5 GHz by choosing appropriate values of Δ (delay) among the various groups.

- Coupling between PCB antennas and digital interfaces should be carefully studied for the overall performance of a PCB.

- Interaction between the return currents of digital interfaces and antenna current profiles in ground planes is the main contributor to coupling. This allows for a successful component placement.

- In receive mode, the crosstalk from the digital interface to the antenna reduces the sensitivity of the receiving module.

- In transmit mode, the crosstalk from the antenna to the digital interfaces deteriorates the eye diagram and leads to higher bit error rates.

- Designs can be simulated and optimized by placing digital interface components outside regions of crowded current distribution and path discontinuities.

- Coupling caused by digital buses on PCB antennas can be reduced by about 50% by applying an appropriate phase shift between different bytes.

- A field solver augmented by CUDA acceleration can be used to solve for fields for large problem sizes to identify potential design problems.