EMC Design Rule Checking – Past, Present, and Future

Sam Connor
sconnor@ieee.org

Distinguished Lecturer for the IEEE EMC Society 2012-13

IBM Systems & Technology Group, Research Triangle Park, NC
My Background

- BSEE, University of Notre Dame, 1994
- Lockheed Martin Control Systems, Johnson City, NY
  - 1994-1996
  - Systems Engineer
- IBM, Research Triangle Park, NC
  - 1996-Present
  - Timing Verification
  - Logic Verification
  - Signal Quality Analysis
  - EMC Design
    - Simulation
    - EMC Design Rule Checker development
    - Research collaboration
Location
Outline

- Past
  - History of EMC
  - Purpose of design rules
  - EMC fundamentals
    - Current
    - Impedance
  - EMC design rules
  - Evolution of design practices

- Present
  - Rule checking choices
  - Required capabilities
  - Considerations

- Future
  - Computing paradigm shift
  - ASIC design rules
  - System-Level rules

- Summary
History of EMC

- **1900 – 1970**
  - Observation of electromagnetic interference in radios, television, and other communications
    - Causes are motors, engines, radar, power distribution lines, etc.
    - Increase in interference with introduction of transistors, ICs, and computing devices

- **1960s**
  - Military and aerospace limits imposed

- **1979**
  - FCC law limiting radiated emissions

- **1980 - 1990s**
  - Emissions limits for computers, peripherals adopted by CISPR
  - Many other countries adopt European Norm (EN) requirements
History of EMC

- As soon as there were limits to meet and sales were at risk...
  - Increased focus on EMC
  - Collection of “lessons learned”
  - Analytical formulae developed from theory
  - Numerical modeling of simplified structures
  - > Development of design rules

- Late 1990s
  - Emergence of automated rule-checking tools
    - IBM developed internal tool in 1992
  - Formation of Research Consortium at University of Missouri at Rolla (UMR)
History of EMC Rule Checking at IBM

- Emphasis on design specifications (workbooks)
- Emphasis on quality
  - How do you know design guidelines were followed?
- Expertise in electromagnetics
- Numerical modeling of simplified structures
- Knowledge of CAD data
- Expertise in computational geometry algorithms

Research Staff

EDA Team

Culture

EMCEP ca. 1992

Motivation

Rules

Implementation
Goal of EMC Rule Checking

Identify violations of design rules and sort them
- Fix most severe violations
- Minimize risk of failure without overdesigning product

Tell me if my product is going to pass EMI testing

Challenges:
- Cannot simulate entire system with all relevant details
- Extrapolating from local effects on PCB to far-field performance
- Products have unique characteristics
  - Airflow requirements
  - Cables/connectors
  - Materials
Electrical vs Mechanical Design Rules

- Both electrical & mechanical designs are critical for overall system EMC performance
- Automation efforts have been limited to Electrical Rules
  - Circuit boards contain the sources of emissions and the victims of susceptibility
  - Circuit boards are more complex and more time consuming to review manually
    - True for simpler systems
    - Must be revisited for current systems
      - Complex high-performance computing racks
      - Modular, integrated products
  - Most EMC engineers have EE backgrounds
EM Wave Propagation

- Accelerating charge creates a propagating EM wave
  - Acceleration of charge = \( \frac{d^2Q}{dt^2} \)
  - \( I = \frac{dQ}{dt} \), so a time-varying current \( \frac{dI}{dt} \) creates a propagating EM wave

- EMC is about **currents**
Can You Identify the EMC Problem from this Schematic?
Follow the Entire Current Path in 3-D Space
Low Frequency Return Currents Take Path of Least Resistance

\[ Z = R + jwL \]

\[ R \gg jwL \]
High Frequency Return Currents Take Path of Least Inductance

\[ Z = R + jwL \]

\[ jwL \gg R \]
PCB Example for Return Current Impedance

22” trace
10 mils wide, 1 mil thick, 10 mils above GND plane
PCB Example for Return Current Impedance

Shortest return path

Longer return path (current returns under trace)
Current Path Impedance Example for U-shaped 22” Trace

- **Z for Shortest Path (L=491nH)**
- **Z for Longer Path (L=10.7nH)**
- **Trace Resistance (One Way)**
- **Inductance Only Longer Path (L=10.7nH)**
- **Inductance Only Shortest Path (L=491nH)**
Surface Current Distribution/Animation at High Frequency (3 GHz)

Type = Surface Current (peak)
Monitor = h-field (f=3) [1]
Component = Tangential
Maximum-3d = 45.0181 A/m at 8922.8 / 41.4 / 1077.2
Frequency = 3
Phase = 0 degrees
Surface Current Distribution/Animation at Mid Frequency (360 MHz)

Type: Surface Current (peak)
Monitor: h-field (f=0.360) [1]
Component: Tangential
Maximum-3d: 40.8293 A/m at 1077.2 / 41.4 / 1077.2
Frequency: 0.361
Phase: 0 degrees
Surface Current Distribution/Animation at Low Frequency (2 MHz)

Type: Surface Current (peak)
Monitor: h-field (f=0.002) [1]
Component: Tangential
Maximum 3D: 34.8311 A/m at 1077.2 / 41.4 / 1077.2
Frequency: 0.002
Phase: 0 degrees
Can You Identify the EMC Problem from this Schematic?

No. We need to know the full current path, which is dependent on layout and frequency.
EMC PCB Design Rules

- Examples of EMC design guidelines
  - Don’t cross splits in reference planes
  - Don’t route nets too close to the edge of a reference plane
  - Bury clock nets and other high energy sources
  - Put filters on I/O lines near the connector
  - Place decoupling near IC power pins
  - Use spatial decoupling to avoid lower-frequency power plane resonances

- List grows with lessons learned
  - Importance of root-cause analysis and feedback

- New rules are needed as technology evolves (higher frequencies, move toward differential signaling, etc)

- Some rules lose importance over time
Signal Reference Rules

- Critical nets must not cross a split in the adjacent reference plane.

- Critical nets must not change reference planes.

- Critical nets must not be within a specified distance of the edge of their reference plane.
Splits in Reference Plane

- Boards with multiple power planes often have splits
  - Sometime crossing cannot be avoided
  - Return current path is interrupted
- Stitching capacitor required across split to allow return current flow
  - Must be close to crossing point
  - Consider stitching capacitor impedance
    - Inductance dominates
- Frequency spectrum of signal is important
  - Clock signals (energy at high harmonics)
  - High frequency harmonics return through displacement current in dielectric
Split Reference Plane Example
Split Reference Plane Example
With Stitching Capacitors

Stitching Capacitors allow return current to cross splits
Capacitor Impedance

Measured Impedance of .01 uf Capacitor

Determined by connection inductance of capacitor
Frequency Domain Amplitude of Intentional Current Harmonic Amplitude
From Clock Net

![Graph showing frequency domain amplitude of intentional current harmonic amplitude from clock net.](image-url)
Microstrip Current Distribution Example – Method of Moments Simulation (100 MHz Clock)
Microstrip Current Distribution Example – Zoomed View
Near Field Radiation from Microstrip on Board with Split in Reference Plane

Comparison of Maximum Radiated E-Field for Microstrip With and without Split Ground Reference Plane

![Graph showing comparison of maximum radiated E-field for microstrip with and without split ground reference plane. The x-axis represents frequency in MHz, ranging from 10 to 1000, and the y-axis represents maximum radiated E-field in dBuV/m, ranging from 20 to 120. Two lines are shown: one for No-Split (blue) and one for Split (pink). The split line shows a significant increase in E-field at higher frequencies.]
With “Perfectly Connected” Stitching Capacitors Across Split

Comparison of Maximum Radiated E-Field for Microstrip
With and without Split Ground Reference Plane and Stitching Capacitors
Stitching Caps with Via Inductance

Comparison of Maximum Radiated E-Field for Microstrip With and without Split Ground Reference Plane and Stitching Capacitors

- No-Split
- Split
- Split w/ one Cap
- Split w/ Two Caps
- Split w/One Real Cap
- Split w/Two Real Caps

Frequency (MHz) vs. Maximum Radiated E-Field (dBμV/m)
Example of Common-Mode Noise Voltage Across Split Plane Vs. Stitching Capacitor Distance to Crossing Point

Gap Voltage

Distance (mils)

0 200 400 600 800 1000 1200 1400 1600 1800 2000

100MHz
200MHz
300MHz
400MHz
500MHz
600MHz
700MHz
800MHz
900MHz
1000MHz
Are Stitching Capacitors Effective?

- It depends
  - Yes, at low frequencies
  - No, at high frequencies
- Limit the high frequency current spectrum
  - Slew rate control
- Avoid split crossings with ALL high-speed (high data rate and/or fast rise time) signals
Wiring and Crosstalk Rules

- Critical nets must not be routed within a specified distance from an I/O net
- Critical nets must be buried between solid planes
- Critical nets must be isolated from other nets by a specified distance
Differential Wiring and Mode Conversion Rules

- Differential vias must have symmetrically placed return vias

- Differential critical nets must be routed within a specified distance of each other, and the length of the mates must match within a specified amount (running skew)
Decoupling Rules

- Decoupling capacitors must be placed between all adjacent plane pairs within a specified grid density (spatial decoupling).

- A decoupling capacitor must be placed within a specified distance from each IC power pin.

- The trace connecting between a capacitor (or IC) pin and its via to the power/ground-reference plane must be no longer than the specified distance.
Placement Rules

- Filters must be placed close to the I/O connector pins they are filtering.

- Certain devices must be placed a specified distance away from other devices or from I/O connectors:
  - Analog and digital isolation
  - Prevent coupling of ASIC emissions to I/O signals
Evolution of Design Practices

- Technology evolves
  - Higher frequencies (data rates and rise times)
  - Higher density (smaller components, better process control)
  - CAD program enhancements
- Design rules must adapt
  - Update models & analytical formulae
  - Build new test vehicles
Design Evolution Example 1

- 10 years ago, all nets were routed with straight lines
  - Also mostly at fixed angles (multiples of 22.5 degrees)
- Now, arcs are supported
  - Requires more advanced algorithms for bounding box and intersection calculations
Design Evolution Example 2

- De-emphasis or change in focus of decoupling rules
  - 10-15 years ago, spacing between power and ground planes was typically 10 mils or more
    - Smaller layer count
    - Fewer devices and lower pin count connected to planes
    - More energy content between 30-300MHz
  - Now, power planes have separations of 3-4 mils (or smaller with special dielectrics)
    - More capacitance between planes reduces dependence on decoupling caps
    - Higher frequency content does not excite the lower-order resonant modes where caps are effective
    - Higher device and pin count lowers the Q of plane resonances

Dielectric thickness dropped from 10 to 4 mils = ~8 dB drop in impedance
Design Evolution Example 3

- Shrinking features, such as pin pitch, create new challenges
  - When antipads overlap, they create a slot in reference plane
  - If CAD data does not join antipads, then rule-checking tool must do it
Design Evolution – Other Examples

- Vias
  - Back-drilling
  - Blind
  - Shared antipad

- Nets
  - Linked nets (i.e. through DC blocking caps or series resistors)

- Layers
  - Embedded capacitance (non-uniform dielectric through stackup)
  - Power/ground shapes on signal layers
Summary

- PCB design for EMC is all about **currents**
  - Pay attention to return current path
- Design rules are needed to achieve EMC compliance
- Main EMC design rules are well known
  - Numerous publications & presentations
  - Use simulation and measurement results to select meaningful limits
- As technology changes, rules need to adapt
Present State of Automated EMC Design Rule Checking
Rule Checking Choices

- Automated vs Manual Reviews
- Develop vs Purchase
- Level of analysis
Progression of EMC PCB Design Rule Checking

- From light tables and transparencies to CAD reviews
  - Rule checking findings migrate from hardcopy to softcopy
- From co-located teams to global teams
  - Harder to conduct manual reviews and communicate issues to designer
- From low layer count and low density boards to high layer count and high density boards
  - Too complex to review manually within time constraints
- From manual reviews to automated reviews
Automated Reviews

- **Pros**
  - Saves analysis time
  - More repeatable
  - Less prone to human error
  - Psychological factor (facts vs opinions)

- **Cons**
  - Have to learn a new tool
  - Development or purchase expense
  - “If we buy a tool, they won’t need me”
Develop vs Purchase

- Develop
  - Have resources to develop algorithms and software
  - Need something special
    - Custom-tailored to your design process
    - Conversion or pre-processing of CAD data

- Purchase
  - Need a solution today
  - Don’t have mix of CAD and software skills available
  - Flexible design process
  - NB: Some tools allow customization
Purchase Options

- The major CAD vendors have a rule checking solution
  - Pros:
    - Integrated in design tool
    - Easy to adopt
  - Things to check:
    - Can it process boards from other vendors?
    - Is the tool supported and maintained by knowledgeable EMC engineers?
    - Can the rules and limits be customized?
    - Cost

- 3rd Party Vendors
  - Pros:
    - Rule checking is main focus, not a supplementary tool
    - Supported and maintained by knowledgeable EMC engineers
    - Support multiple CAD formats
  - Things to check:
    - Can the rules and limits be customized?
    - How easily will the tool fit into your design process?
    - Cost
Level of Analysis

- Simple geometrical design rule check
  - Very fast
  - Straight-forward to use and interpret

- “Expert System” analysis
  - Moderate speed
  - More complicated calculations
  - Attempts to provide more guidance on whether to fix a problem and how
  - Requires understanding of assumptions and limitations
Rule Check vs. Expert System Example

- Avoid Exposed (Microstrip) Traces
  - Rule Check
    - Set a limit on total exposed length
  - Expert System
    - Calculate field strength at 3m/10m away based on radiation from a microstrip
      - What if 5cm are exposed on each end? Does radiation add?
      - What if the PCB has a shield around it?
      - What frequency(ies) are calculated?
Rule Checking Capabilities

- Rule configuration
- Design classification
- Results
  - Reports
  - Visualization of violations
Rule Configuration

Tool must provide an intuitive way to:

– Define your “playlist”
  ▪ Which rules to run
– Define your rule limits
– Define which options are enabled
  ▪ Adjustments to how the rule works
– Store and recall settings
Design Classification

- Tool must provide an intuitive way to define which nets and components are important
  - Automatically classify by naming convention
  - Manually classify by selection within graphical interface (CAD tool)
  - Manually classify in a spreadsheet-like interface

**Control File:**
- CPU* = CPU Nets
- *CLK* = Clock Nets
- DDR* = Mem Nets
- P*_DD* = PCIe Nets

**Net Name**

<table>
<thead>
<tr>
<th>Net Name</th>
<th>Clock Nets</th>
<th>CPU Nets</th>
<th>Mem Nets</th>
<th>PCIe Nets</th>
</tr>
</thead>
<tbody>
<tr>
<td>CPU_D01</td>
<td>X</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CPU_CLK_P</td>
<td>X</td>
<td>X</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DDR_DQ01</td>
<td></td>
<td></td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>P01_DD01</td>
<td></td>
<td></td>
<td></td>
<td>X</td>
</tr>
</tbody>
</table>
Results

- Tool must provide a way to review violations
  - Convenient report format that can be sent to team members
    - Standard formats are usually best (HTML, PDF, XLS, ODF, etc)
  - Visual display provides best context for decision making
EMC Design Rule Checking Tasks/Workflow

- **Rule configuration (1-2 minutes)**
  - Setup limits and options to suit the product technology
  - Not a per-product effort
    - Asynchronously adjusted as an ongoing research effort

- **Design classification or “tagging” (2-4 hours)**
  - Schematics, net lists, experience
  - Boolean expressions and wildcard string matching to find nets & components

- **Execute rules (1-60 minutes)**
  - Repeat for all rule configurations necessary

- **Review violations (2-4 hours)**
  - View violations with context (in design view)
  - Look for patterns (repeated design features)
  - Prioritize violations
    - Number of fixes is often limited
Optimized EMC Design Rule Checking Workflow

- Reduce time required to classify design
  - Enforce naming conventions on design teams
    - Tagging can be done automatically with scripts (1-2 minutes)
- Reduce time required to review violations
  - Apply post-processing algorithms to violation list
    - Eliminate commonly ignored violations based on recognizable messages or patterns
Rule Checking Application Considerations

- **Usability**
  - Does it have to run inside the CAD environment or can it be a separate process?
  - How does the user review output?

- **Portability**
  - Support for multiple operating systems

- **Maintainability**
  - Use a modern language (balance of what skills are present in your organization and availability of compilers, libraries, etc)
  - Object-oriented design

- **Modularity and Expandability**

- **Measurability**
Modularity and Expandability

- Easy to add new rules
  - Adapt to technology changes
  - Incorporate other rulesets
    - Signal Integrity
    - Power Integrity
    - Thermal? Mechanical? Other?
- Easy to adjust settings and limits for rules
  - No recompiling or modifying of scripts required
- Support for multiple CAD formats
Measurability - Process Improvement through Analytics

- Continual process improvement
  - Needs feedback loop
- Benefits:
  - Measurable improvements to tool
  - More effective usage of tool

1. Collect Performance Data
2. Run Rule Checks on PCB
3. Adjust Rules and/or Settings
4. Monitor Average Performance vs Target
Measurability - Monitor Average Performance

Good profile (green circle)
- Green and red bars close to the same, with blue bar somewhat higher (conservative checking)

Bad profiles (red circle and red arrows)
- Green and red bars close to zero, with blue bar much higher
Summary

- Automate!
  - Speed
  - Consistency
- When selecting a tool or developing one
  - Be wary of “expert” tools that apply algorithms beyond their scope
  - Choose flexibility
    - Ease of adding and customizing rules
  - Consider integration with design process
    - Setup, rule execution, and results evaluation
    - Tracking
Future of Automated EMC Design Rule Checking
Computing Paradigm Shift

- **Cloud computing**
  - **Advantages:**
    - Tools pre-loaded and pre-configured in a virtual instance
    - Shared rule and design resources
    - Offload computationally intensive analyses
    - Enables usage tracking
  - **Issues:**
    - Graphics performance over web interface
    - Licensing for CAD tools
    - Intellectual property security concerns
Design Rule Checking for ASIC Packages

- Organic packages are small PCBs
- Packages are electrically large above 3 GHz (10cm wavelength)
- With each new silicon technology family:
  - Smaller gate sizes
  - Faster slew rates
  - Higher emissions
Previous Work on EMC of ICs

- Measure near-fields above IC
  - Find design patterns that cause “hot-spots”
  - Convert to equivalent dipole sources and predict far-field radiation
- Issues:
  - Can you measure near-fields with lid on?
  - What if the IC needs a heatsink to operate with normal traffic?
  - Helps with system-level simulations and design, but usually too late to impact IC design
Common Threads in Recent Literature

- PCB-Package Co-Simulation
  - Include package parasitics in end-to-end link simulations
  - Marry PDN characteristics of both domains to get total picture of power delivery to chip

- SI-PI Co-simulation
  - Include power integrity effects in signal integrity simulations

- Where is EMI in this discussion?
  - EMI tends to be ~20dB more sensitive than SI
Rules to Explore

- Decoupling rules for package
  - Via stitching and decoupling between planes
  - Adequate number of BGAs between PCB and package for power and ground nets
- PCB wiring rules applied to package
  - Signal referencing
    - Splits
    - Reference plane changes
  - Signals buried and away from edges of planes
  - Differential Pair Skew
- Lid grounding
- Other?
Design Rule Checking for Systems

- Most existing rule checking tools and efforts focus on PCBs
- Many issues found in the lab are caused by mechanical or system integration issues
  - Missing or ill-fitting gaskets
  - Cables and connectors between PCBs
  - Grounding
Design Rule Checking for Systems

- Extend PCB rules to multi-board scenarios (running skew, signal reference continuity, I/O filter placement)
Design Rule Checking for Systems

- Work on ways to process mechanical CAD files and identify holes, slots with missing gaskets, other
Challenges of Mechanical Rule Checks

- Supporting multiple CAD formats
- Modularity
- Tolerances
- Metal coatings
- 3D vs “2.5D”
Design Rule Checking for Systems

- Combine electrical and mechanical design data and evaluate grounding, excessive coupling between parts
Design Rule Checking for Systems

- Develop way to visualize problems for easy reviewing
  - Standard formats
    - STEP
    - 3D-PDF
    - U3D
  - Embedded in CAD tool with scripts
Summary

- Time to move beyond checking individual PCBs
- Cloud computing opens new possibilities
- Rule-checking at IC package level
  - IC packages are small PCBs
  - ICs are not electrically small above 3-5 GHz
  - Many PCB design rules apply directly
- Rule-checking at System level
  - Many EMC issues are related to system integration
    - Check mechanical features
    - Check electrical-to-electrical, electrical-to-mechanical, and mechanical-to-mechanical interfaces