FTF | FREESCALE TECHNOLOGY FORUM 2015

EMI/EMC Design for Automotive and Industrial Controls

FTF-DES-F1304

Prof. Todd H. Hubing | Clemson University

JUNE.2015

External Use

EMC Requirements and Key Design Considerations

Radiated Emissions

- 1 HF GND
- Risetime Control
- Filtered I/O
- Adequate Decoupling
- Balance Control

Radiated Susceptibility

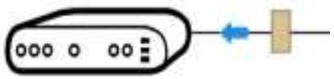
- 1 HF GND
- Filtered I/O
- Adequate Decoupling
- Balance Control

Transient Immunity

- LF Current Path Control
- Chassis GND on board
- Filtered I/O
- Adequate Decoupling

Electrostatic Discharge

- LF Current Path Control
- Chassis GND on board
- Filtered I/O
- Adequate Decoupling



Bulk Current Injection

- 1 HF GND
- Chassis GND on board
- Filtered I/O
- Adequate Decoupling
- Balance Control

Designing a product that is guaranteed to meet all of these requirements is relatively easy. Fixing a non-compliant product can be difficult and costly.

Agenda

- Identify Your Ground Structure
- Recognize Where Currents Flow
- Control Your Transition Times
- Recognize the 4 (not 2) Possible Coupling Mechanisms
- Identify Your Antennas
- Identify Your Sources
- Don’t Rely on EMC Design Guidelines
- Use the Right Shield for the Right Application
- Provide Adequate Power Bus Decoupling
- Provide Adequate Transient Protection

Automotive and Industrial Design for EMC

To guarantee that your design will meet its EMC requirements the first time:

- **IDENTIFY**
  Your ground structure
- **TRACE**
  Your current return paths

Definition of Ground

ANSI C63.14 - 1992 Dictionary for Technologies of Electromagnetic Compatibility

4.151 - Ground, Facility System

The electrically interconnected system of conductors and conductive elements that provides multiple current paths to earth. The facility ground system includes the earth electrode subsystem, lightning protection subsystem, signal reference subsystem, fault protection subsystem, as well as the building structure, equipment racks, cabinets, conduit, junction boxes, raceways, ductwork, pipes, and other normally non-current-carrying metal elements.

4.152 - Grounding

(1) The bonding of an equipment case, frame, or chassis to an object or a vehicle structure to ensure a common potential.

(2) The connecting of an electric circuit or equipment to earth or to some conducting body of relatively large extent that serves in place of earth.

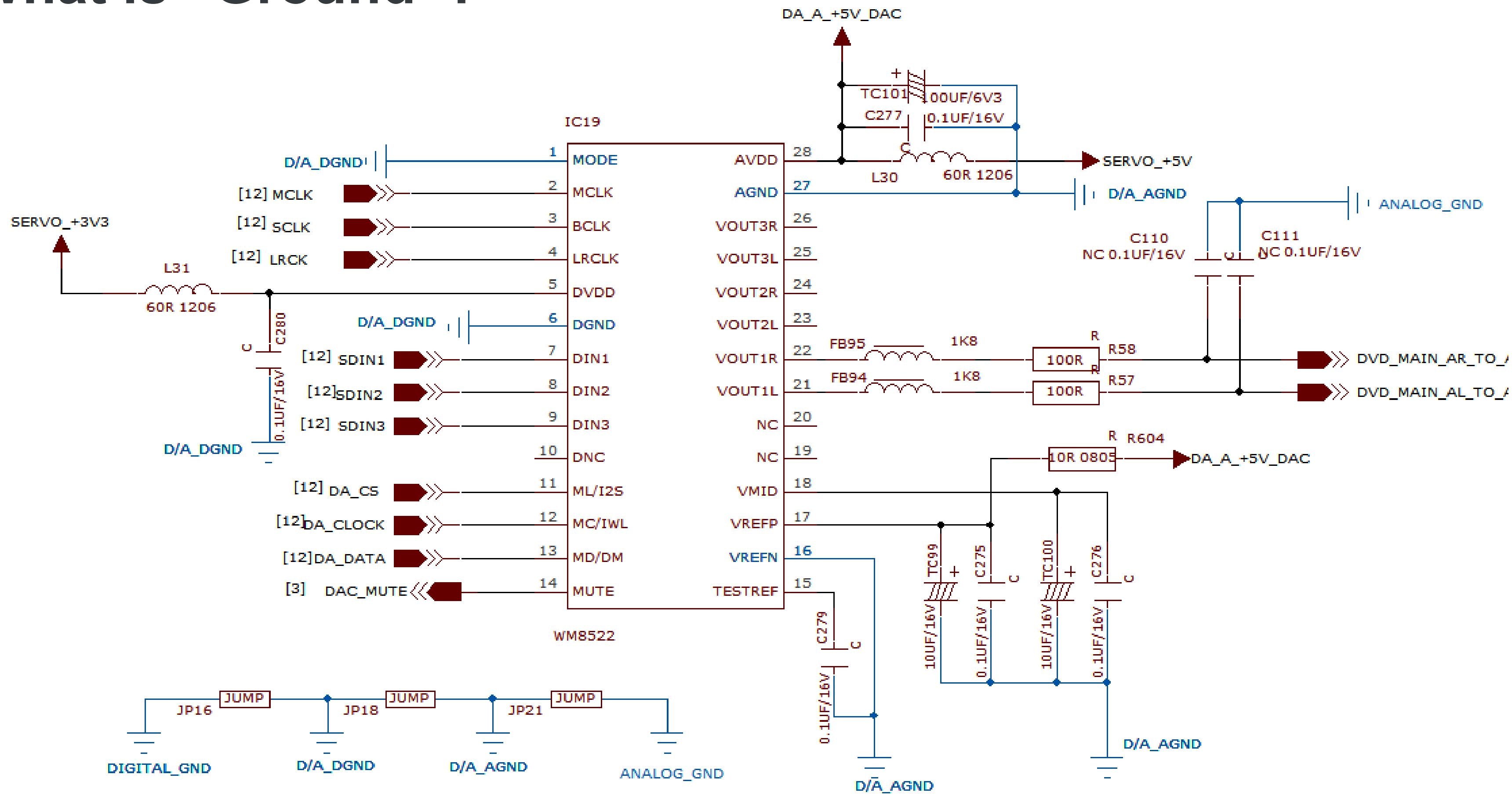
National Electrical Code

Ground

A conducting connection, whether intentional or accidental, between an electrical circuit or equipment and the earth or to some conducting body that serves in place of the earth.

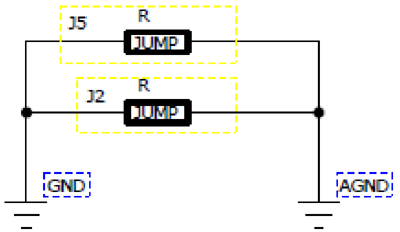
Ground is a conductor that serves as a reference potential and does not carry current!

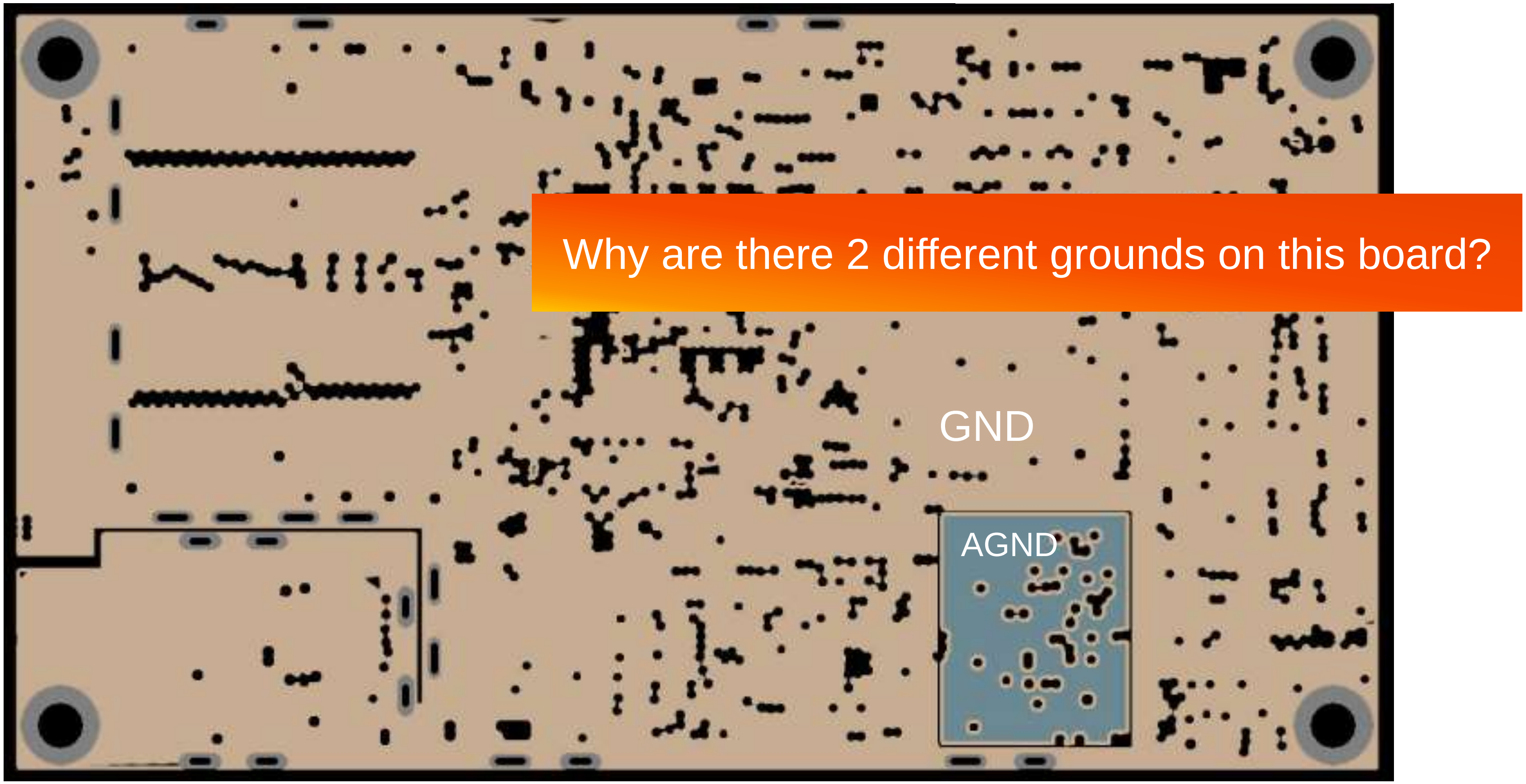
What is “Ground”?



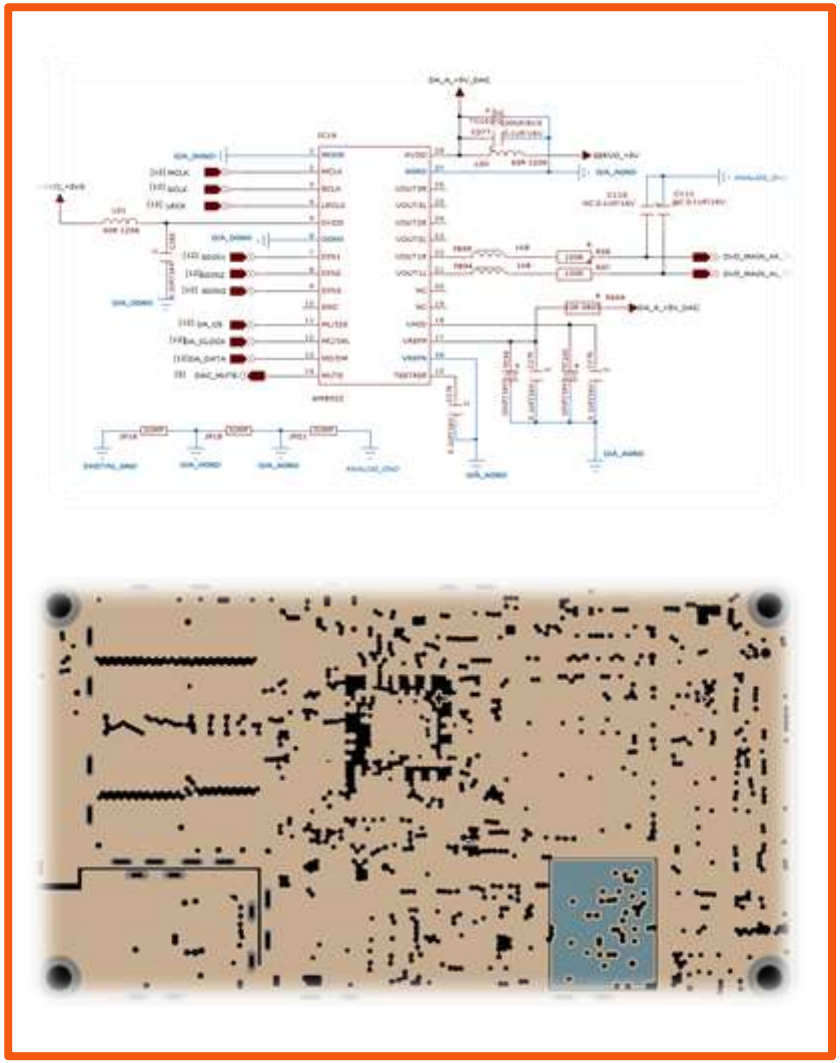
Why are there 4 different grounds in this circuit?

What is “Ground”?





What is "Ground"?



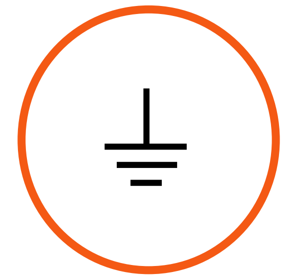
These are not grounds!

They are current return conductors!

They are isolated to prevent Common-Impedance Coupling!

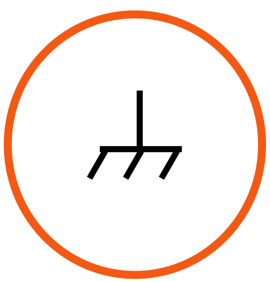
Grounding is not the same as current return!

Ground vs. Signal Return



"Whenever I see more than one of these symbols on the schematic, I know there is [EMC] work for us here."

T. Van Doren



Current Returns vs. Ground Conductors

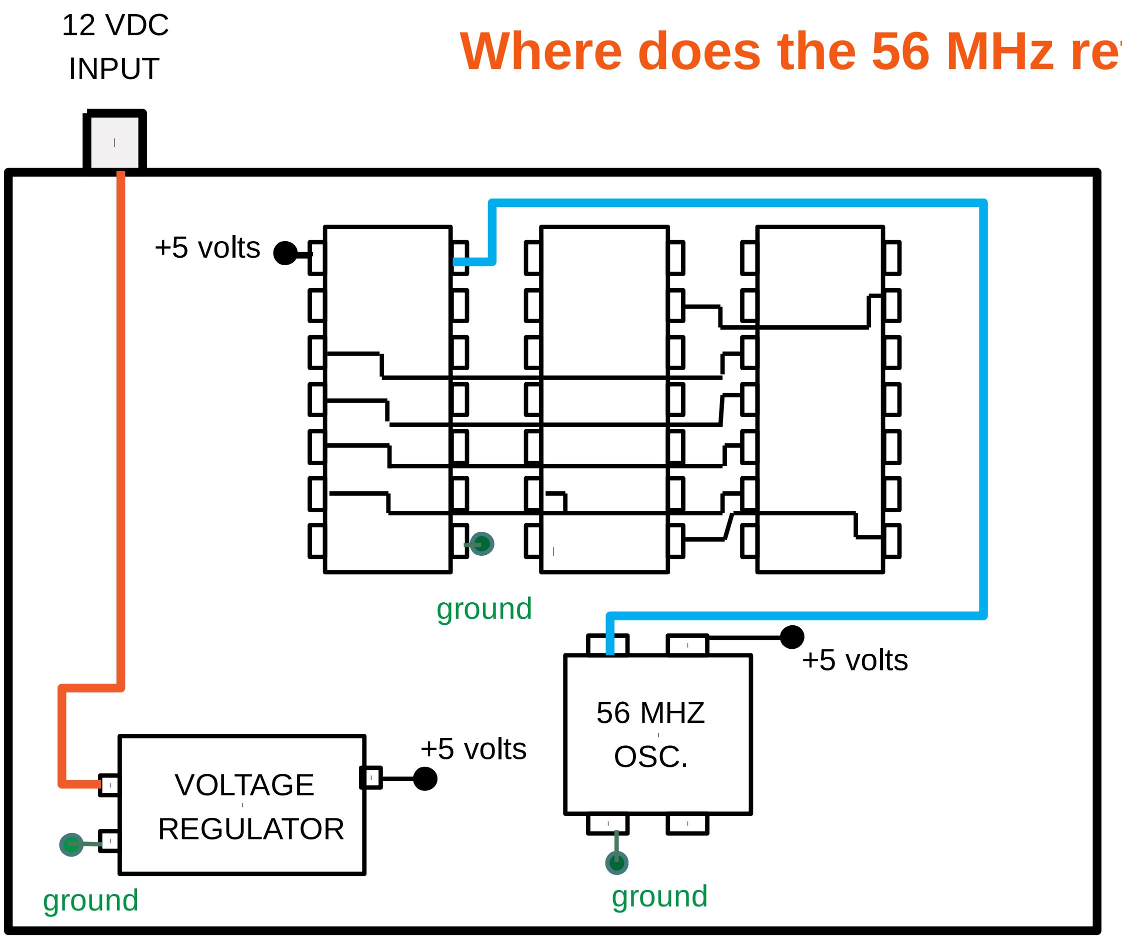
- Two of the most important skills necessary to design systems that meet all electromagnetic compatibility requirements:
  - Ability to identify the current return paths for power and signal currents
  - Ability to differentiate between a current return conductor and a ground conductor

Identify Current Paths

- **Current takes the path of least impedance!**
  - > 100 kHz this is generally the path of **least inductance**
  - < 10 kHz this is generally the path(**s**) of **least resistance**

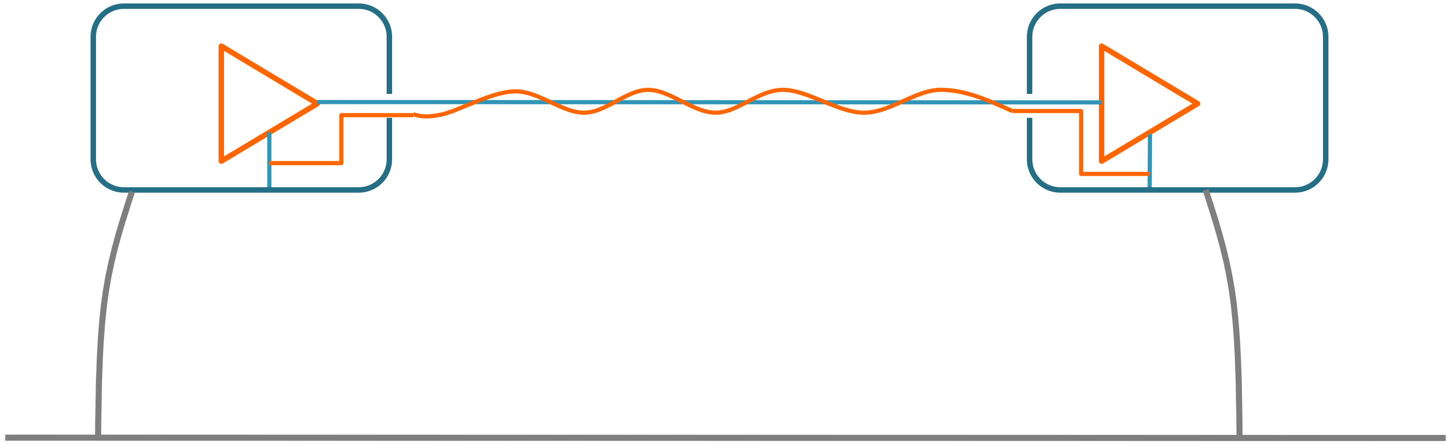
Identify Current Paths

Where does the 56 MHz return current flow?



Identify Current Paths

Where does the return current flow?

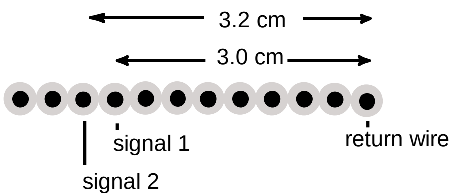


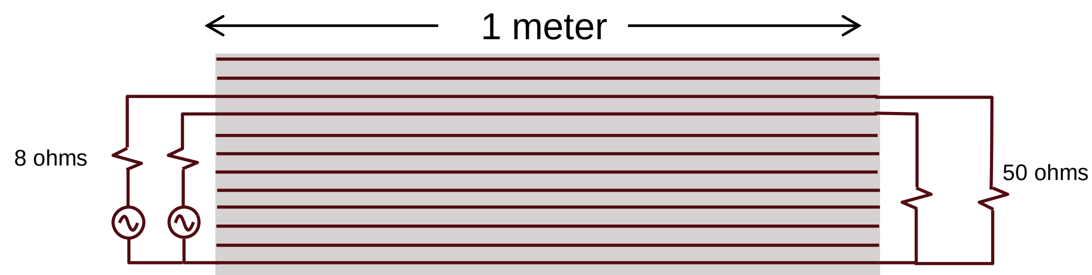
Low-frequency currents often “prefer” to return through the building/chassis rather than our designated return path.

Identify Current Paths

Where does the 10 MHz return current flow?

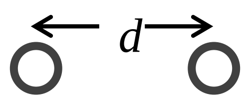
24 AWG





Parallel Wires

$$L = \frac{\mu}{2\pi} \cosh^{-1}\left(d/a \right)$$



wire radius: a

At 1 MHz:

$$L = 1.65\ \mu H$$

$$\omega L = 10.4\ \Omega$$

At 10 MHz:

$$L = 1.65\ \mu H$$

$$\omega L = 104\ \Omega$$

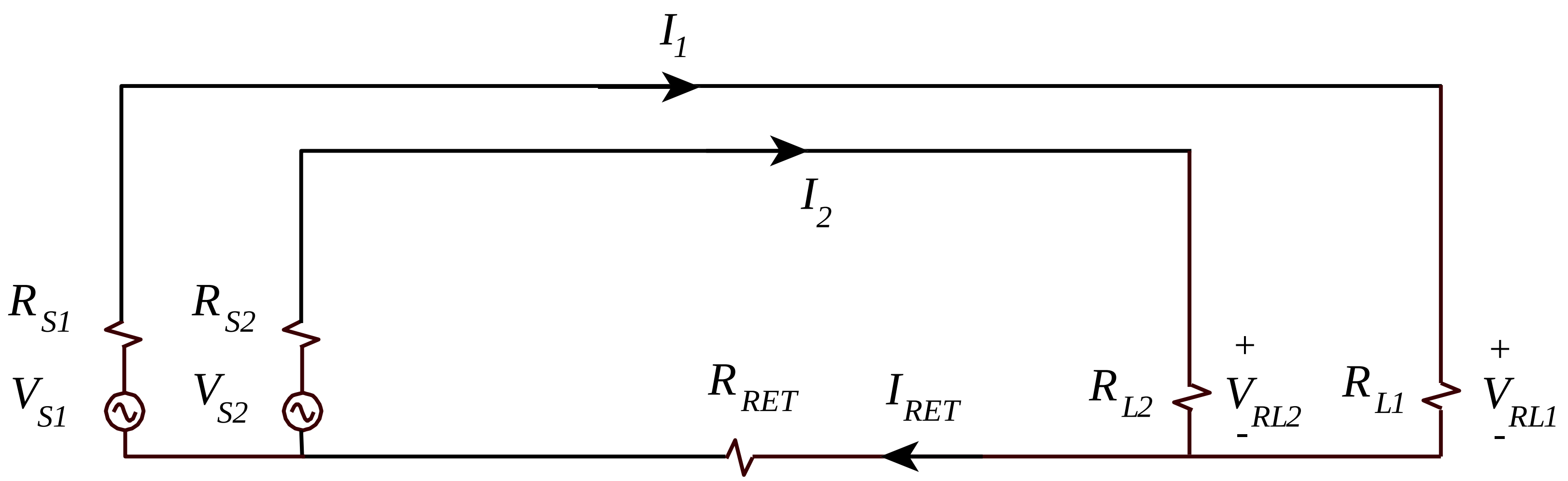
High-frequency currents often “prefer” to return through the nearest conductor rather than our designated return path.

Key Points

- Above 1 MHz, all we have to do is provide a good low-inductance return paths for all signals and the currents will take those paths.
- Below 100 kHz, maintain control of current return paths. Don’t allow currents that differ by 1 - 3 orders of magnitude to share the same return path(s).

So is it 1 or 3?

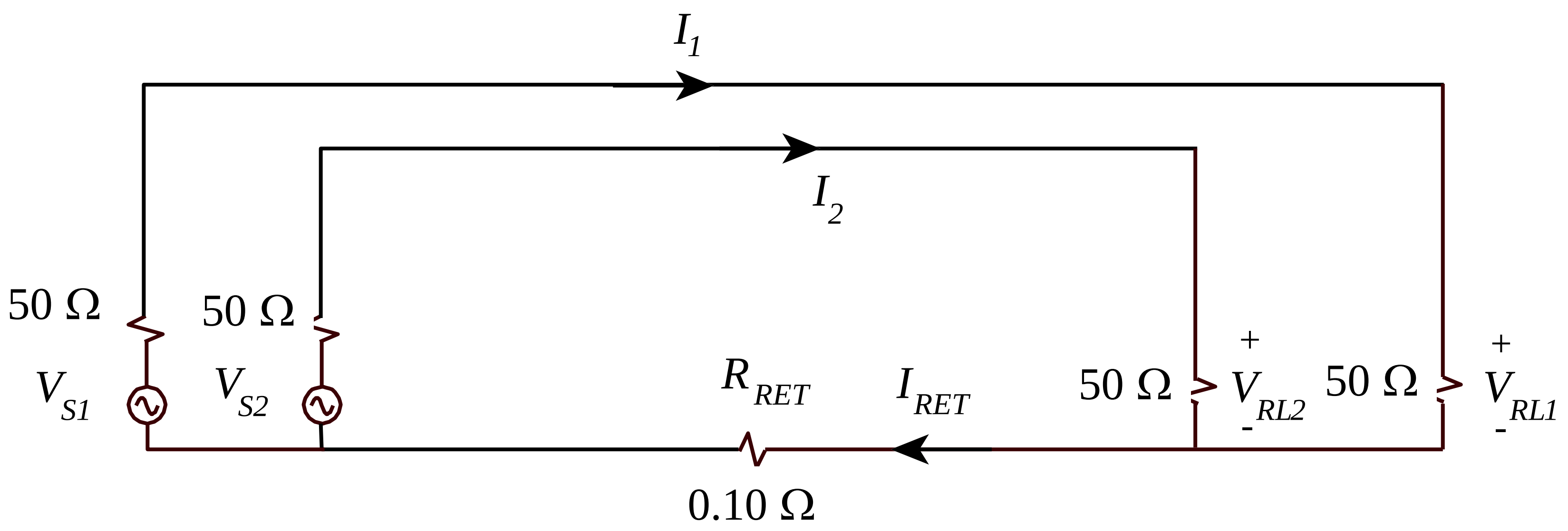
Crosstalk



$$crosstalk\ in\ dB = 20\ log\left|\frac{coupled\ voltage\ appearing\ at\ receiver\ in\ Circuit\ 2}{signal\ voltage\ in\ Circuit\ 1}\right|$$

For the circuits above: $Xtalk_{21} = 20\ log\left|\frac{V_{RL2}}{V_{RL1}}\right|_{when\ V_{S2}=0}$

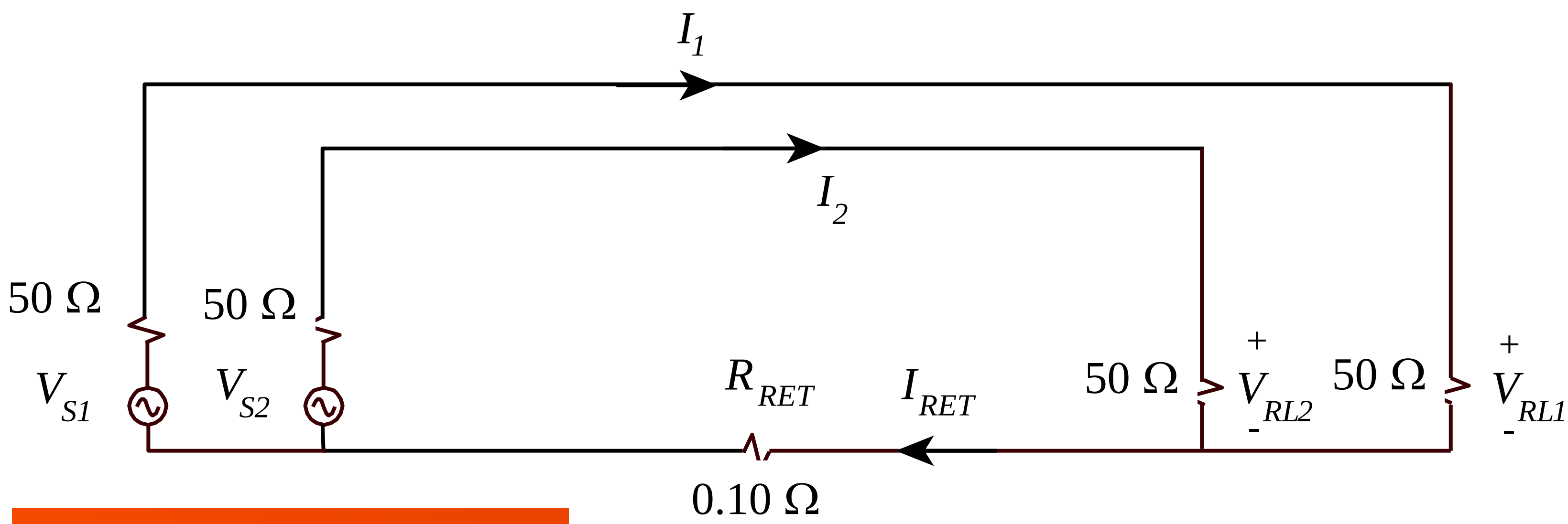
Crosstalk



We can solve this circuit using SPICE or loop and node equations, but that takes time and inhibits an intuitive understanding for what is really happening.

$$Xtalk_{21} = 20 \log \left| \frac{V_{RL2}}{V_{RL1}} \right|_{when\ V_{S2}=0}$$

Let's Solve For the Crosstalk in this Circuit



$$Xtalk_{21} = 20\log\left|\frac{V_{RL2}}{V_{RL1}}\right|_{when\ V_{S2}=0}$$

$$= 20\log\left|\frac{V_{RET}/2}{V_{S1}/2}\right|$$

$$= 20\log\left|\frac{V_{S1}\left(0.1/100\right)}{V_{S1}}\right|$$

$$= -60\,dB$$

Rules for Current Return Routing

$$\text{maximum common impedance} \leq \frac{\text{maximum source current}}{\text{minimum receiver interference voltage}}$$

- Two circuits that operate at voltages or currents that differ by an order of magnitude or more should not share the same return trace or wire.
- At frequencies **below** 1 MHz, two circuits that operate at voltages or currents that differ by more than two orders of magnitude or more should not share the same return plane on a circuit board.
- At frequencies **above** 1 MHz, circuits can share the same return plane on a circuit board provided their currents do not overlap. (Remember, the return currents are confined to the region of the plane immediately below a microstrip trace.)

Rules for Grounding

- So those are the rules for current return.
- What about the rules for grounding?

Definition of Ground

ANSI C63.14 - 1992 Dictionary for Technologies of Electromagnetic Compatibility

4.151 - Ground, Facility System

The electrically interconnected system of conductors and conductive elements that provides multiple current paths to earth. The facility ground system includes the earth electrode subsystem, lightning protection subsystem, signal reference subsystem, fault protection subsystem, as well as the building structure, equipment racks, cabinets, conduit, junction boxes, raceways, ductwork, pipes, and other normally non-current-carrying metal elements.

4.152 - Grounding

(1) The bonding of an equipment case, frame, or chassis to an object or a vehicle structure to ensure a common potential.

(2) The connecting of an electric circuit or equipment to earth or to some conducting body of relatively large extent that serves in place of earth.

National Electrical Code

Ground

A conducting connection, whether intentional or accidental, between an electrical circuit or equipment and the earth or to some conducting body that serves in place of the earth.

System Ground and Ground Conductors

- The purpose of a system ground is to provide a reference voltage and/or a safe path for fault currents.
- In order to serve this function, a ground conductor cannot carry any “objectionable” current.

Ground Conductors vs. Signal Return

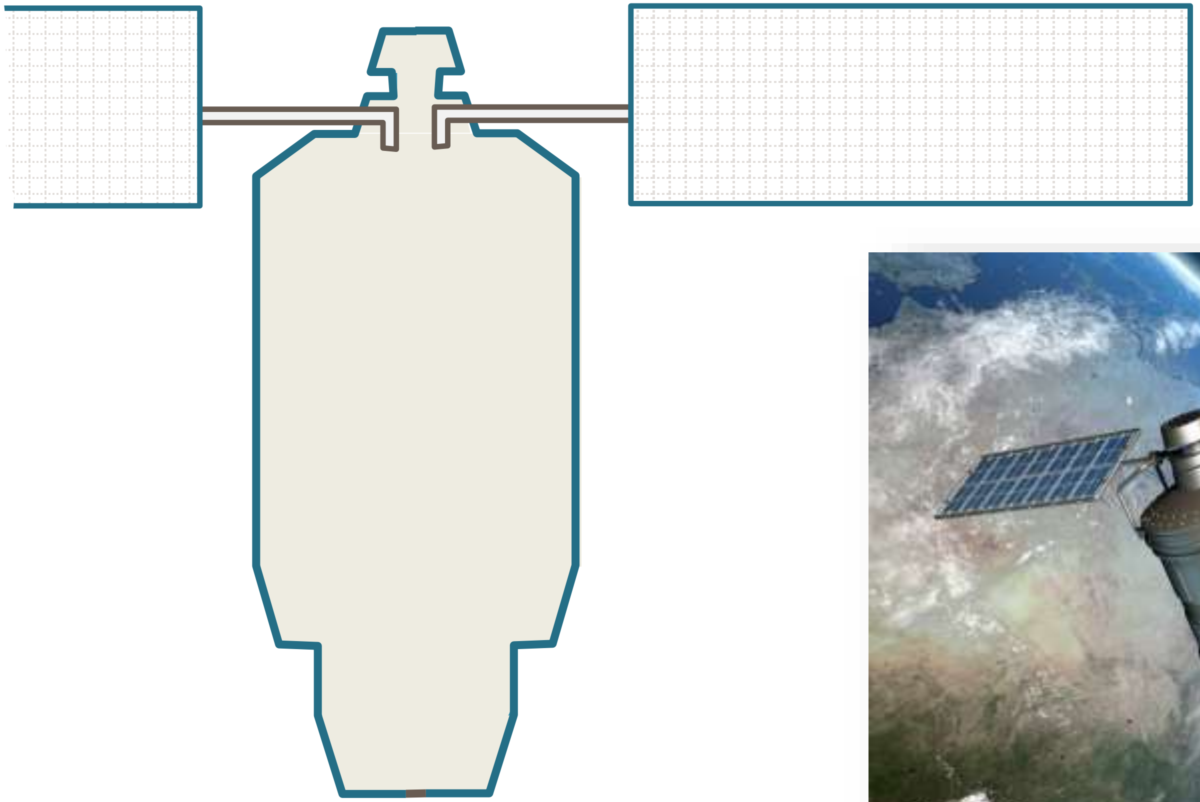
- The purpose of a system ground is to provide a reference voltage and/or a safe path for **fault** currents.
- Signal or power currents flowing on a “ground” conductor can prevent a ground conductor from serving its intended purpose.
- **Don’t confuse ground conductors with signal return conductors.** Rules for the routing of “ground” may conflict with the rules for routing signal or power returns.

Ground Structures

- Serve as the system ground
- Provide a local reference potential throughout the entire system

Ground Structures

- A ground structure doesn't need to be electrically small to be effective.
- However, it is important not to induce a voltage between any two parts of the ground structure.

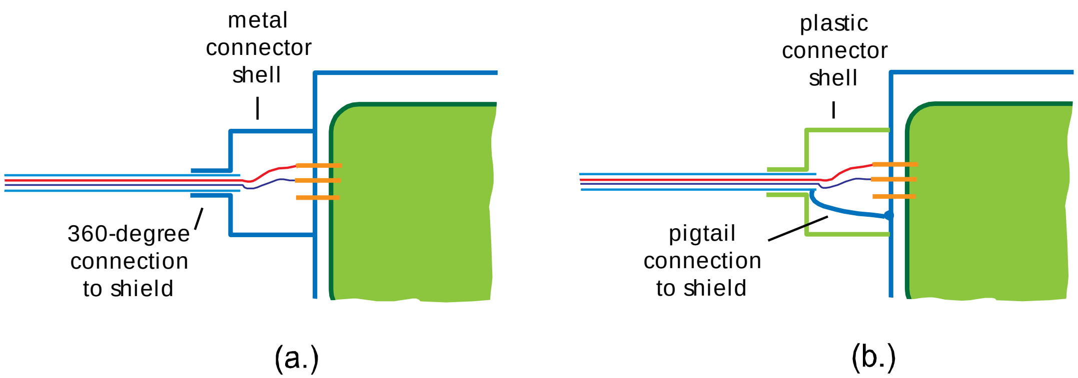


Ground Structures

Ground Structures

- Are good conductors
- Are accessible throughout the system
- May be electrically large
- Do not carry intentional signal currents

Grounding Conductors



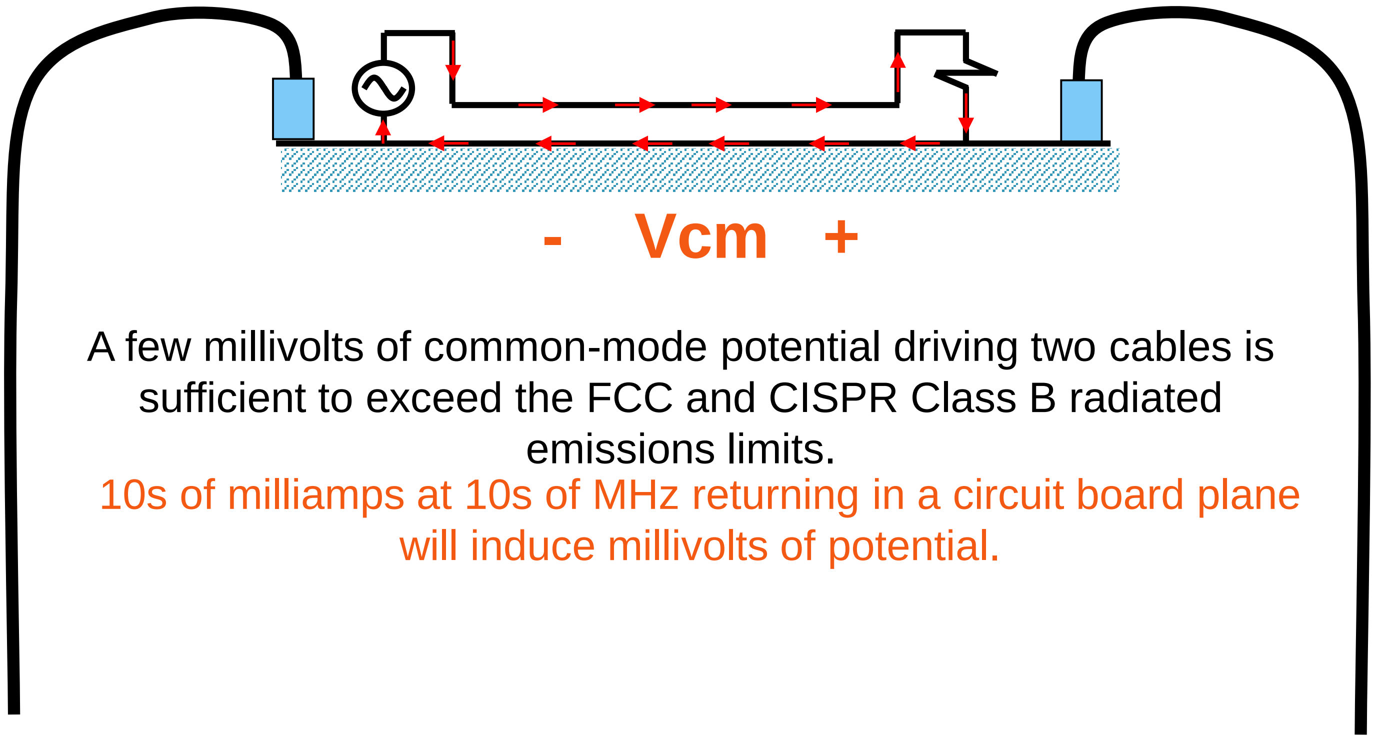
Grounding Conductors

- Are good conductors
- Have low inductance as well as low resistance
- May **not** be electrically large
- Do not carry intentional signal currents

Is a “Ground Plane” a Ground Structure?

Current Driven Radiation Mechanism

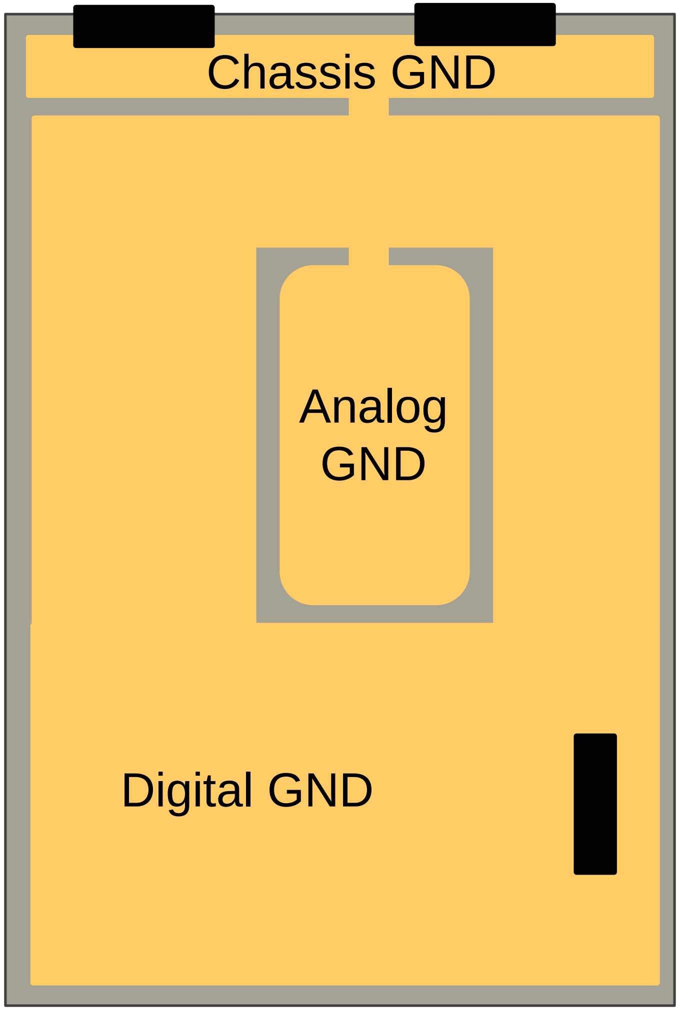
Signal current loop induces a voltage between two good antenna parts.



A few millivolts of common-mode potential driving two cables is sufficient to exceed the FCC and CISPR Class B radiated emissions limits.

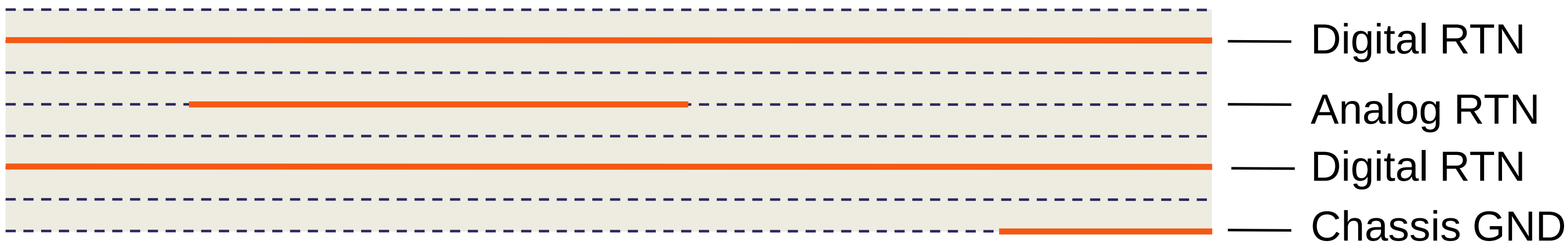
10s of milliamps at 10s of MHz returning in a circuit board plane will induce millivolts of potential.

Where is the “Ground Structure”?



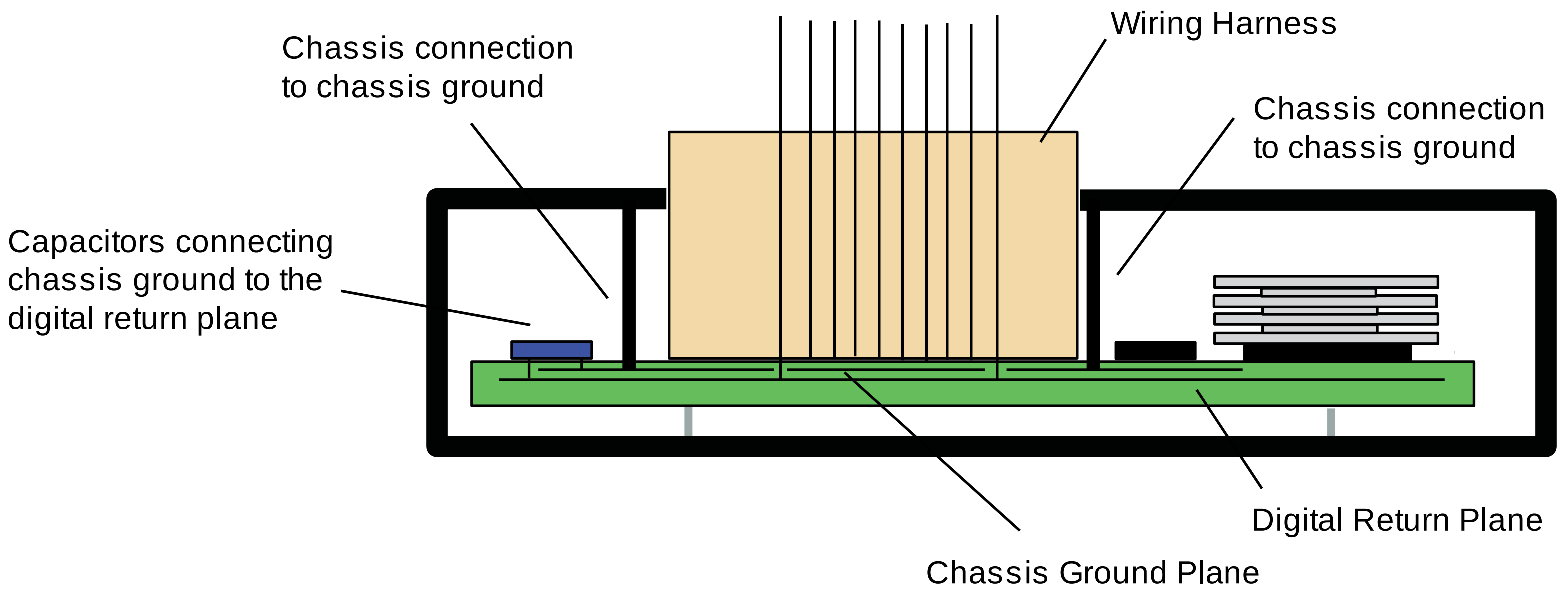
Can we guarantee that each attached cable is within 1 mV of the ground structure at radiated emissions frequencies?

Vertical Isolation



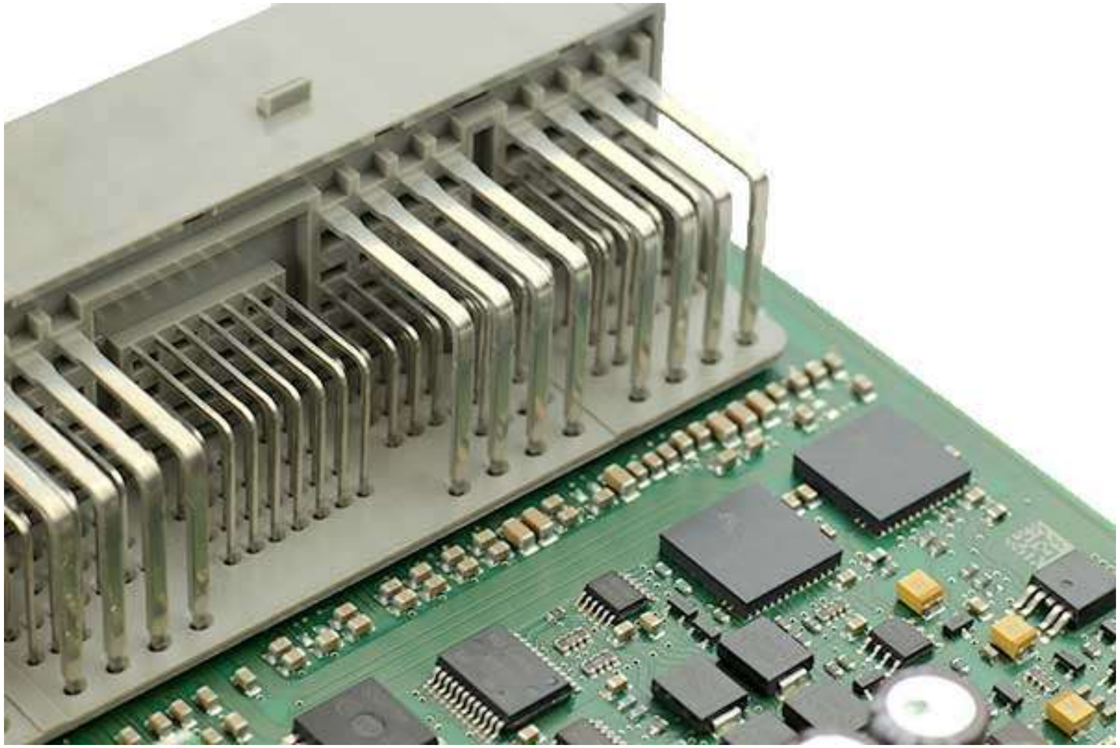
- All planes that reference signals that leave the board should be tied together with capacitors.
- Only one plane usually needs to be full size.
- One or zero vias should connect planes with different labels.

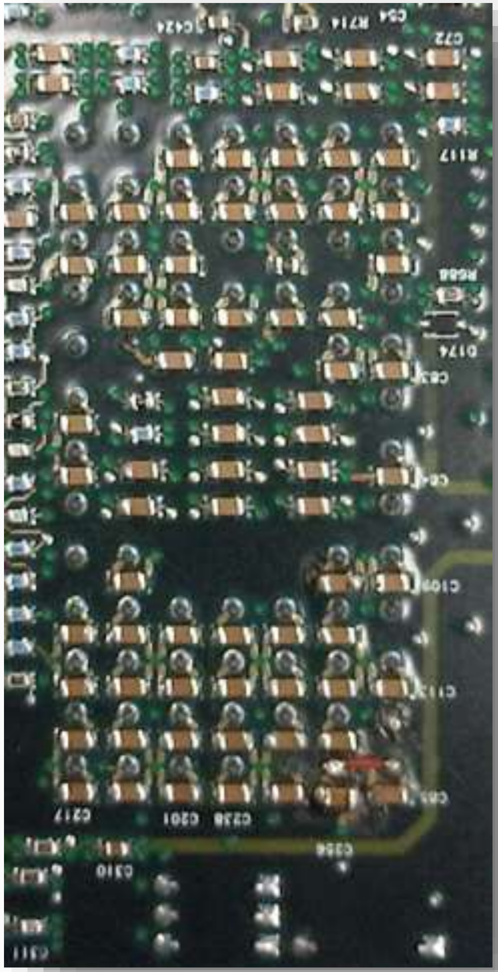
Isolating Chassis and Digital Grounds



Capacitors from I/O to Chassis

Caps on every connector pin





Better implementation

How to Control Radiated Emissions

Circuit boards should have one high-frequency ground!

- And you should be able to be able to identify it without hesitation.
- It can be a grounding structure, or
- in the absence of a grounding structure, it can be a specific location.

Why?

- Conductors referenced to different grounds can be good antennas.

Rules for Grounding

- Designate **one** location or one non-current-carrying metal structure as your zero-volt reference or ground.
- Be sure that all other metal structures including attached cables and large heatsinks do not deviate from the ground potential by more than an acceptable limit.
- For radiated emissions (10s of MHz and higher), this acceptable limit is on the order of 1 mV.
- For safety, the acceptable limit is generally on the order of 10s of volts.

Rules

For Grounding

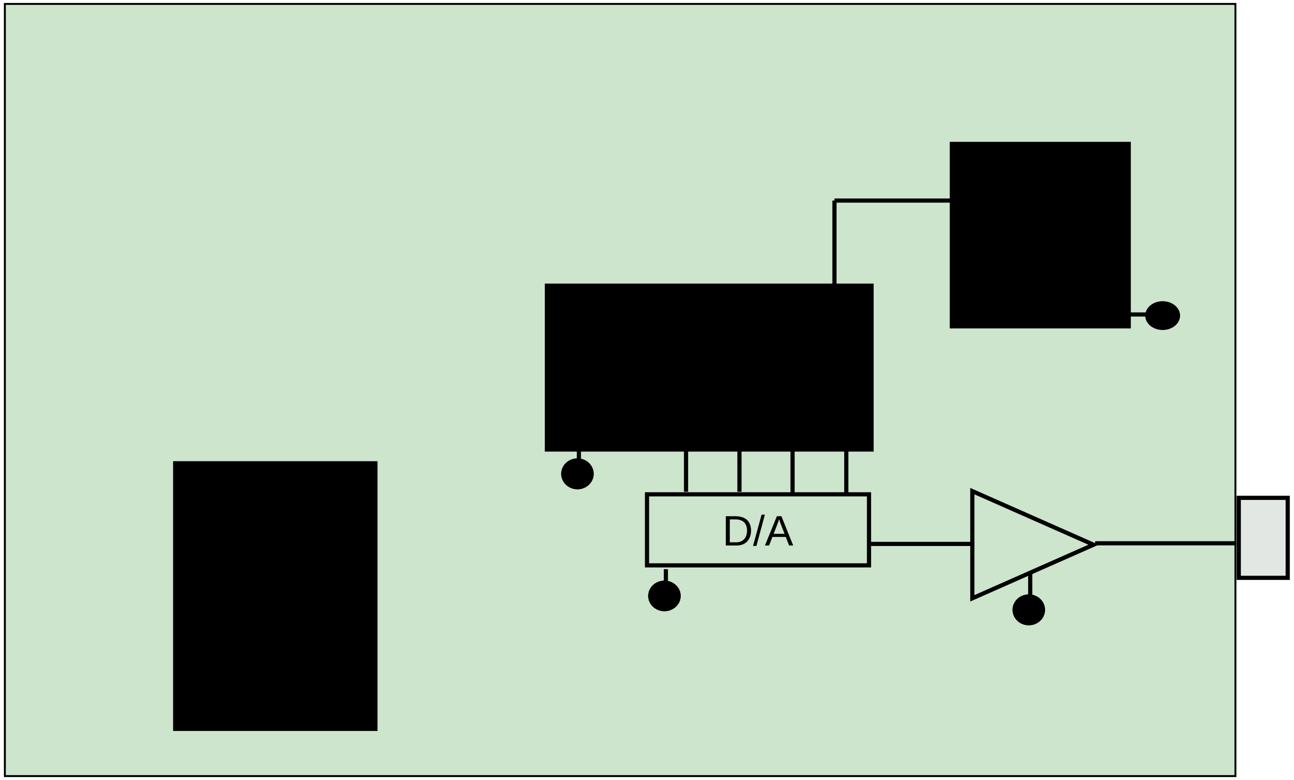
- Designate one location or one non-current-carrying metal structure as your zero-volt reference or ground.
- Be sure that all other metal structures including attached cables and large heatsinks do not deviate from the ground potential by more than an acceptable limit.
- For radiated emissions (10s of MHz and higher), this acceptable limit is on the order of 1 mV.
- For safety, the acceptable limit is generally on the order of 10s of volts.

For Current Return Routing

- Two circuits that operate at voltages or currents that differ by an order of magnitude or more should not share the same return trace or wire.
- At frequencies below 1 MHz, two circuits that operate at voltages or currents that differ by more than two orders of magnitude or more should not share the same return plane on a circuit board.
- At frequencies above 1 MHz, circuits can share the same return plane on a circuit board provided their currents do not overlap. (Remember, the return currents are confined to the region of the plane immediately below a microstrip trace.)

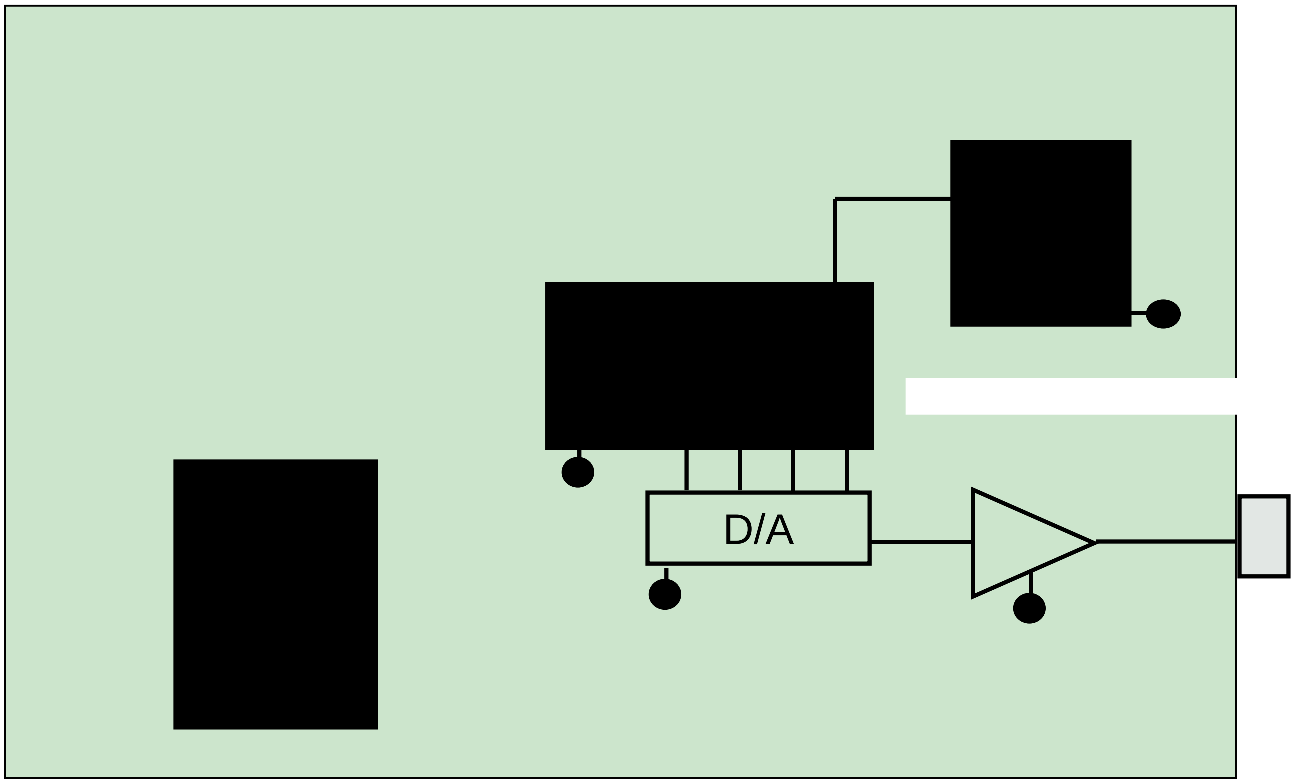
Ground vs. Signal Return

Exercise: Trace the path of the digital and analog return currents.



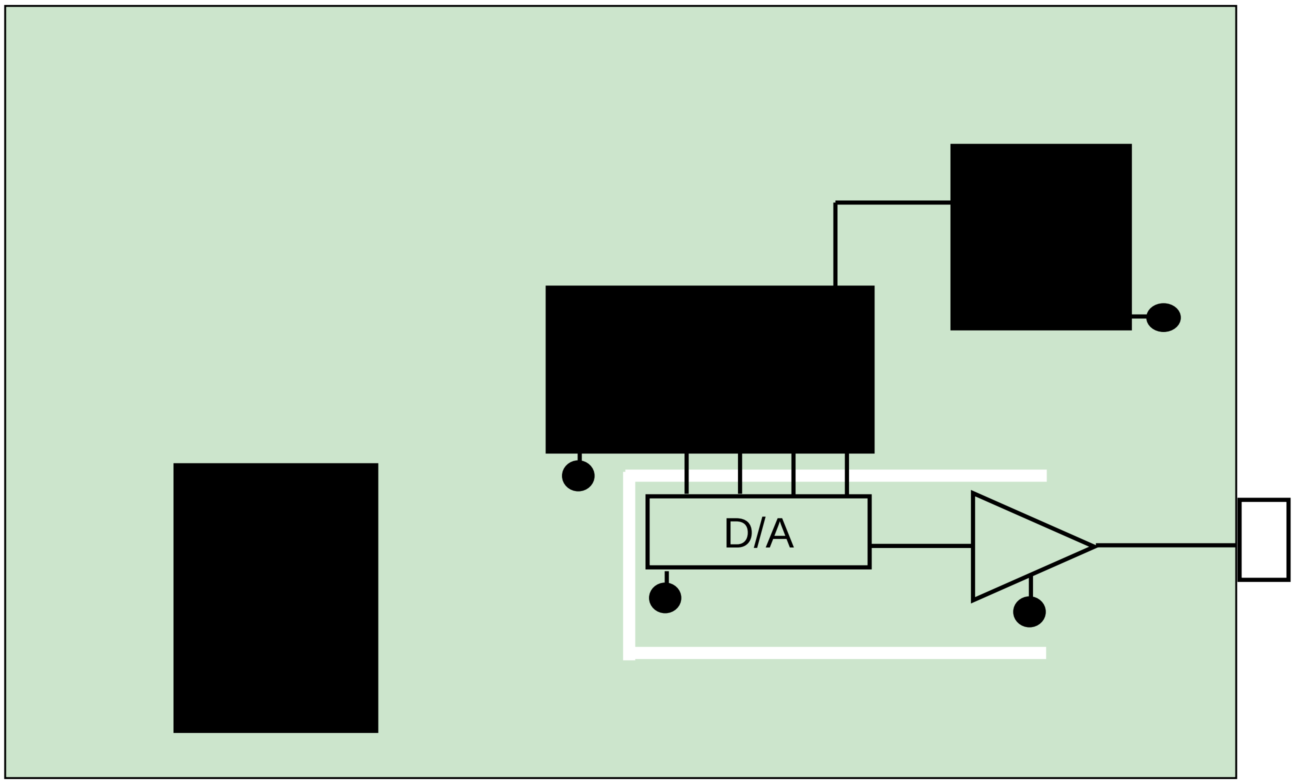
Ground vs. Signal Return

Exercise: Trace the path of the digital and analog return currents.



Ground vs. Signal Return

Exercise: Trace the path of the digital and analog return currents.



Ground vs. Signal Return

You don't need to gap a plane to control the flow of high frequency (>1MHz) currents. If you provide a low-inductance path for these currents to take, they will confine themselves to this path very well.

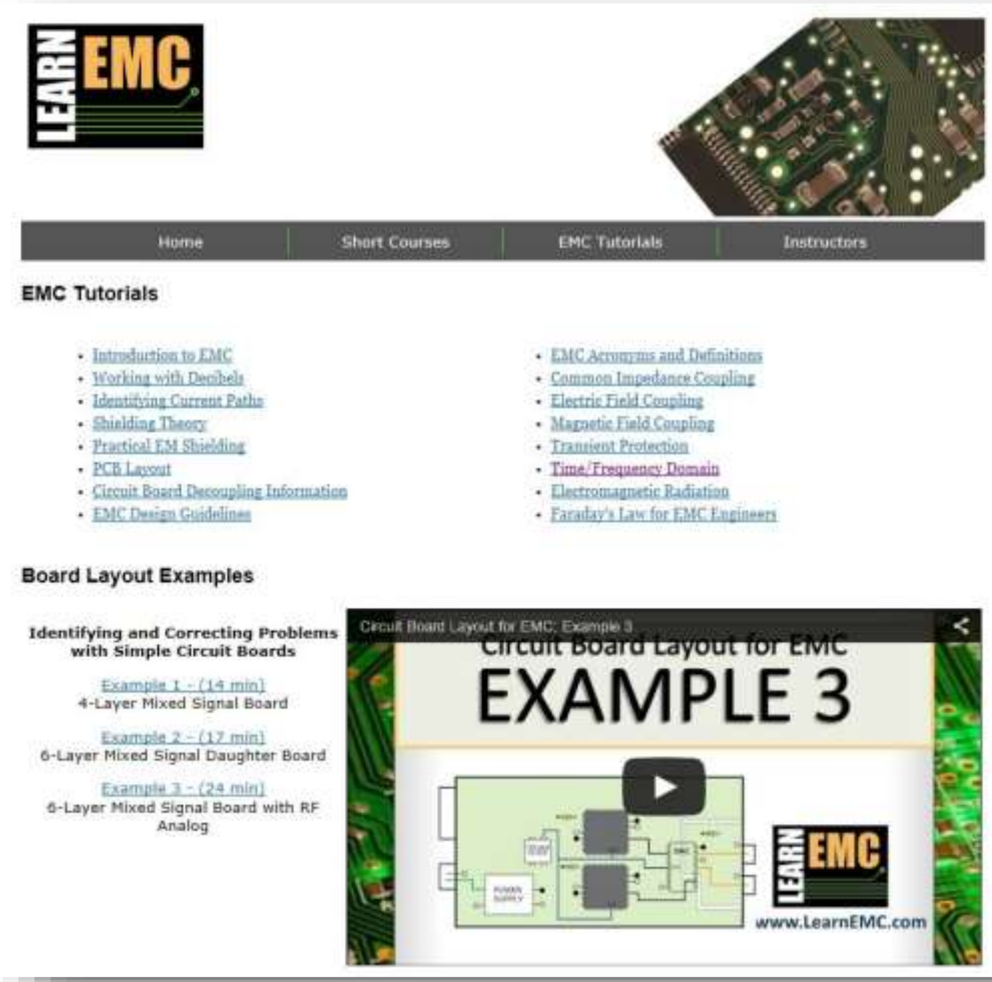
Rules for gapping a ground plane:

- **Don't do it!**
- If you must do it, never ever allow a trace or another plane to cross over the gap.
- If you must do it, never ever place a gap between two connectors.
- Remember that the conductors on either side of the gap are at different potentials.
- See Rule #1!

Ground vs. Signal Return

Hint for working the PCB Design Examples on LearnEMC Tutorials page:

Always eliminate the gap in the ground plane.

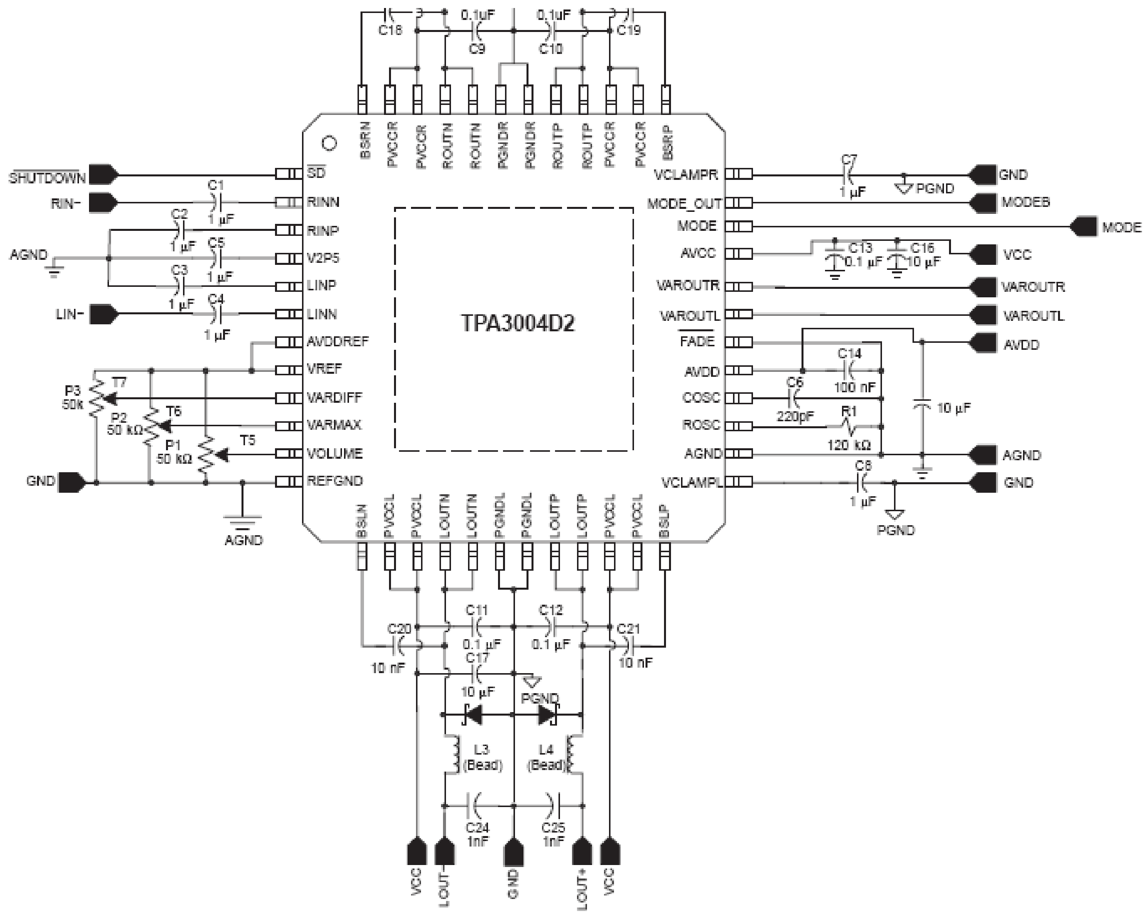


Mixed-Signal Designs

If you have analog and digital returns that must be isolated (to prevent common-impedance coupling):

- Route the returns on separate conductors
- Provide a DC connection at the one point (or in the one area) where the reference potential must be the same.
- This must include every place where a trace crosses the boundary between the analog and digital regions.

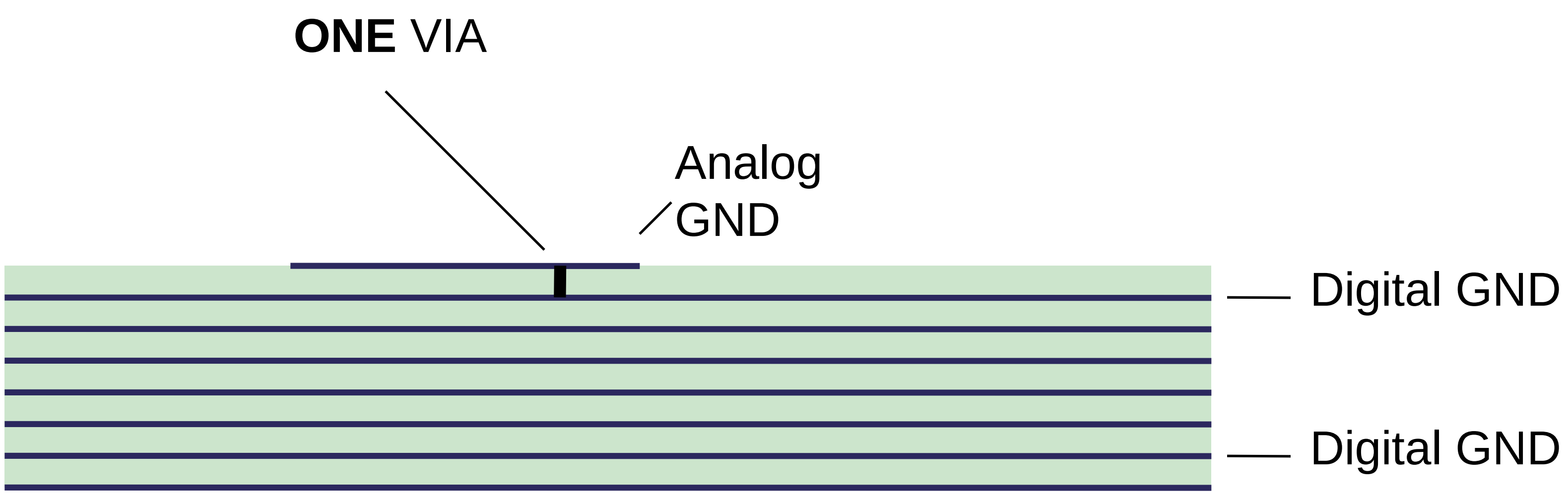
Mixed Signal Designs



†Schottky diodes only needed for short circuit protection when V_{CC} > 15 V. See SHORT CIRCUIT PROTECTION section in Application Information.

Figure 35. Stereo Class-D With Single-Ended Inputs

Sensitive A/D Isolation



If you think you need two vias, then you shouldn't be isolating the analog and digital grounds.

Design Advice

- Identify your HF ground and be sure it is the only ground that is large or connected to anything large!
- Don't call anything other than current carrying nets "ground". For example, refer to a current carrying analog reference net as "analog return".
- Be aware of where your HF and LF currents are flowing!
- Isolate returns only when necessary to control the flow of low frequency currents.
- **If you isolate two large conductors at low frequencies, be sure they are well connected at high frequencies.**

Summary of Key Points

- Grounding is a critical aspect of EMC and product safety.
- Grounding is all about providing a reference potential.
- Grounding is **not** about returning currents to their source.
  - Unfortunately, many current return nets in circuits are labeled ground, and
  - Paying attention to current return paths is also an important aspect of meeting EMC and signal integrity requirements.

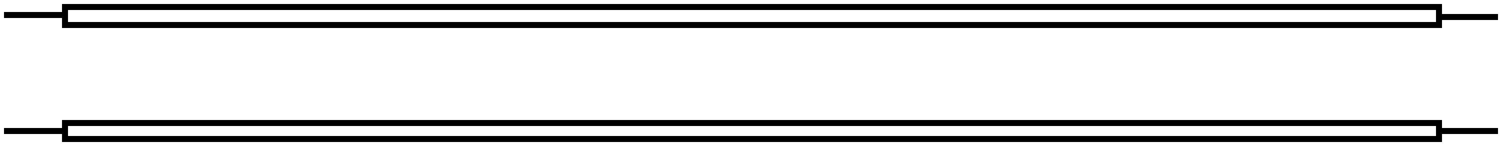
- Identifying and maintaining the integrity of a **grounding structure** is an important part of designing for EMC and product safety.

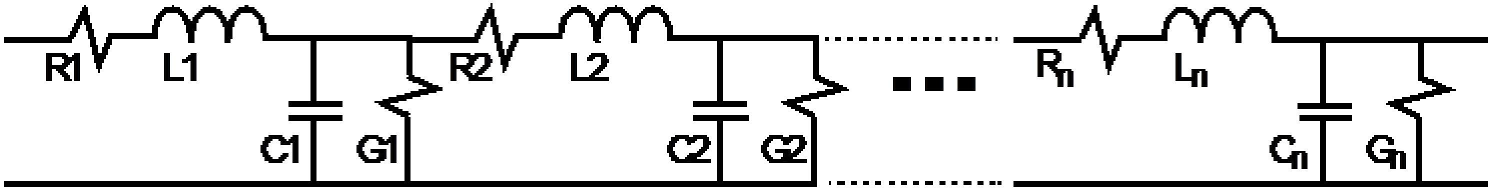
Automotive and Industrial Design for EMC

To guarantee that your design will meet its EMC requirements the first time, you must:

CONTROL
your transition times!

RLCG Parameters





If the length of each section of the RLCG lumped model is small relative to a wavelength (e.g. $l_n \ll \lambda/8$), the electrical behavior of the model is the same as the electrical behavior of the transmission line.

Propagation Velocity

Determined by the dielectric material

Propagation velocity (m/sec)

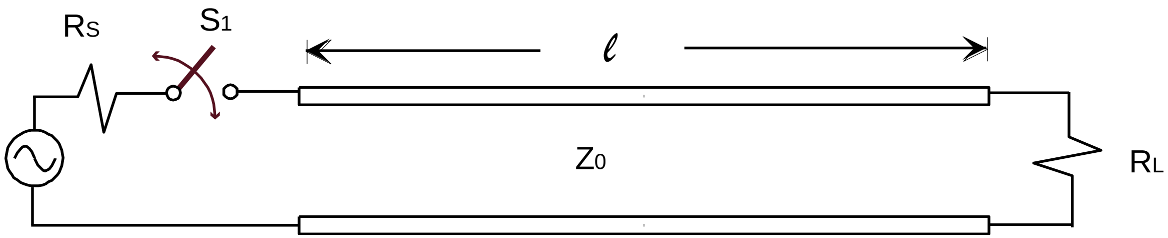
$$v = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{\mu\varepsilon}}$$

Inductance per unit length (H/m)

Capacitance per unit length (F/m)

This term is independent of the geometry

Propagation Delay (Electrical Length)

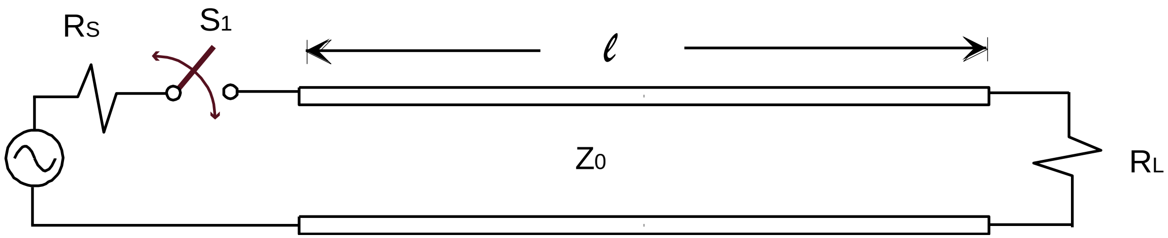


$$t_{PD} = \frac{l}{v}$$

Propagation Delay (sec)

The propagation delay is the amount of time required for a signal to propagate from one point to another point (total distance, l) on the transmission line.

Characteristic Impedance



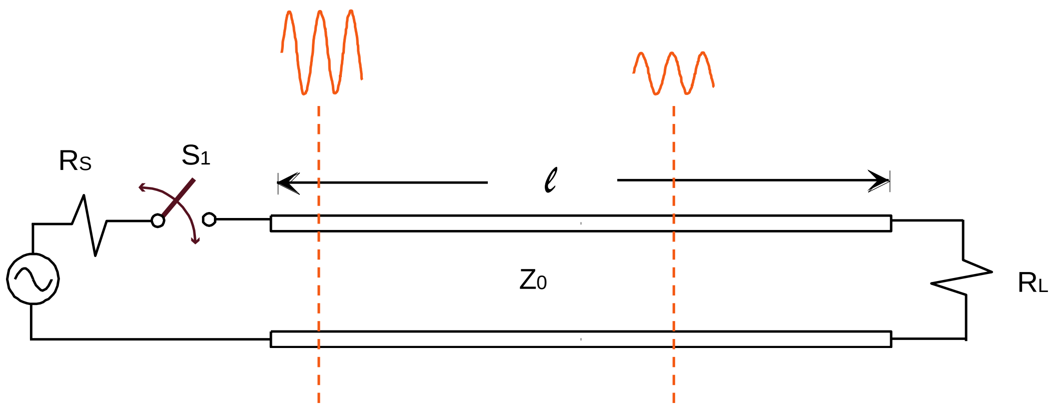
$$Z_0 = \sqrt{\frac{R + j\omega L}{G + j\omega C}} \approx \sqrt{\frac{L}{C}}$$

Characteristic Impedance (ohms)

Low-Loss Approximation

The characteristic impedance is the ratio of the voltage to the current in a signal traveling in one direction down the transmission line.

Attenuation



$$V = V_0 e^{-\gamma x}$$

$$\gamma = \sqrt{(R + j\omega L)(G + j\omega C)} = \alpha + j\beta$$

Attenuation in dB/m $= 20\log\left(\dfrac{V_0 e^{\alpha(x=0m)}}{V_0 e^{\alpha(x=1m)}}\right)$

$= 20\log\left(e^{-\alpha}\right)$

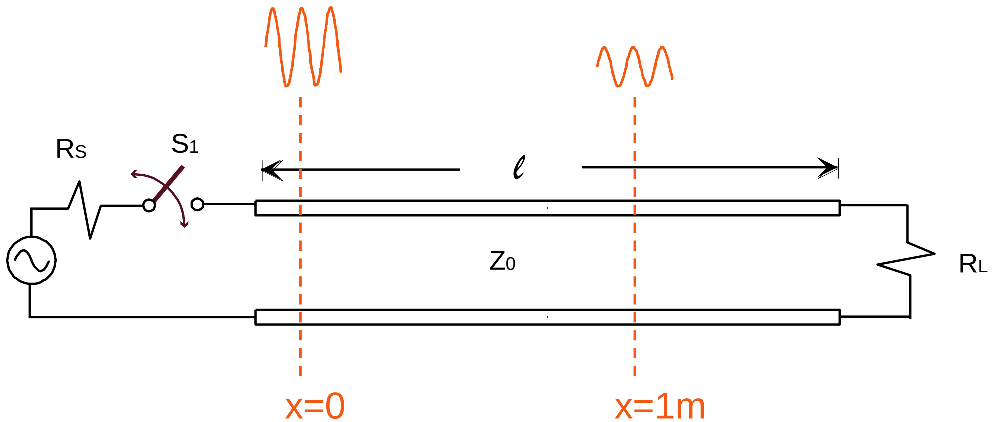
$= 8.7\alpha$

$$\alpha \approx \frac{R}{2Z_0} \qquad \beta \approx \omega\sqrt{LC}$$

Low-Loss Approximation

Attenuation in dB/m $\approx \dfrac{4.34R}{Z_0}$

Dispersion



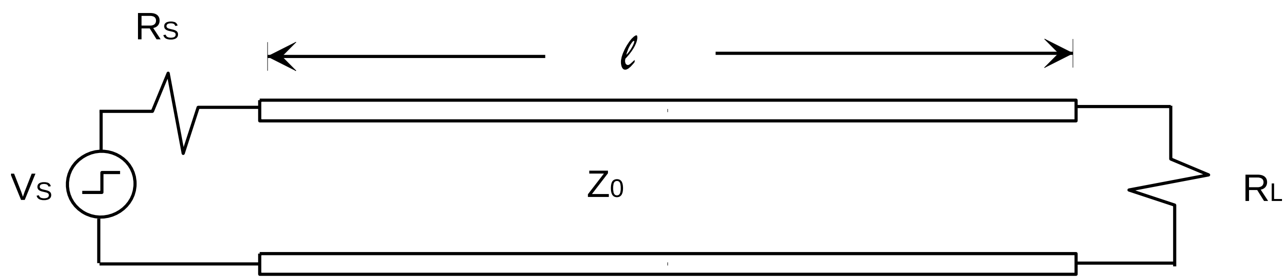
$$\alpha \approx \frac{R}{2Z_0} \qquad \beta \approx \omega\sqrt{LC}$$

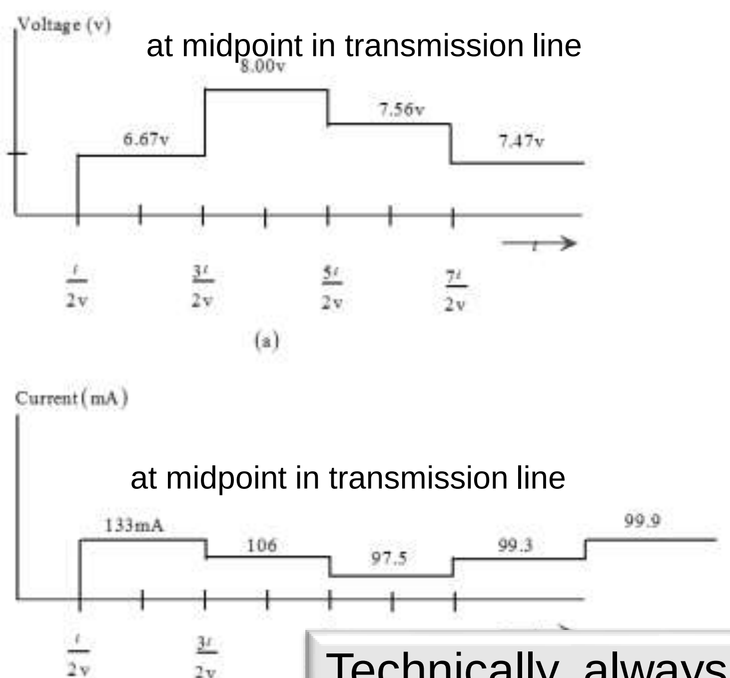
Low-Loss Approximation

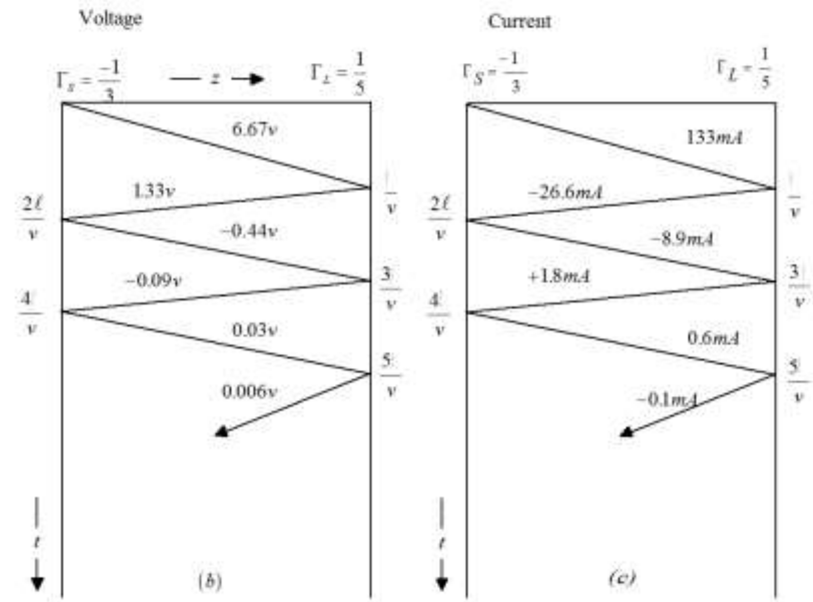
$$\text{Attenuation in dB/m} \approx \frac{4.34R}{Z_0}$$

Notice that attenuation is a function of R, but at high frequencies, R is a function of frequency due to the skin effect. Therefore higher frequencies are attenuated more than lower frequencies. This can change the shape of the signal in the time domain, and this effect is called dispersion.

When is a Cable a Transmission Line?

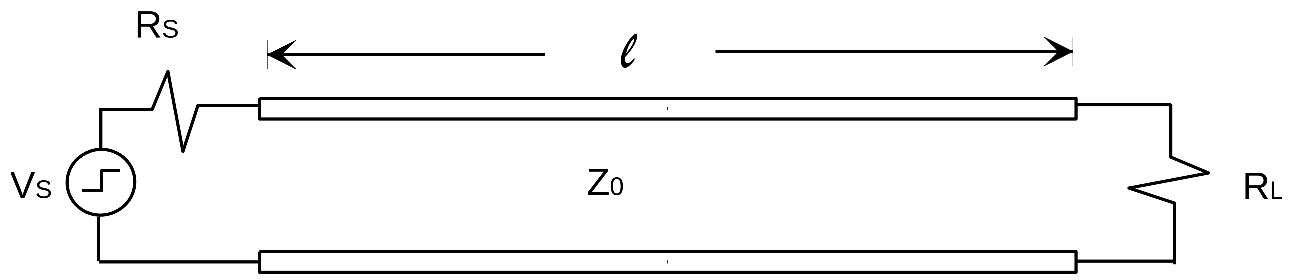






Technically, always! But it will be most important to us when the propagation delay is greater than the transition time.

When Must a Cable be Modeled as a Transmission Line?

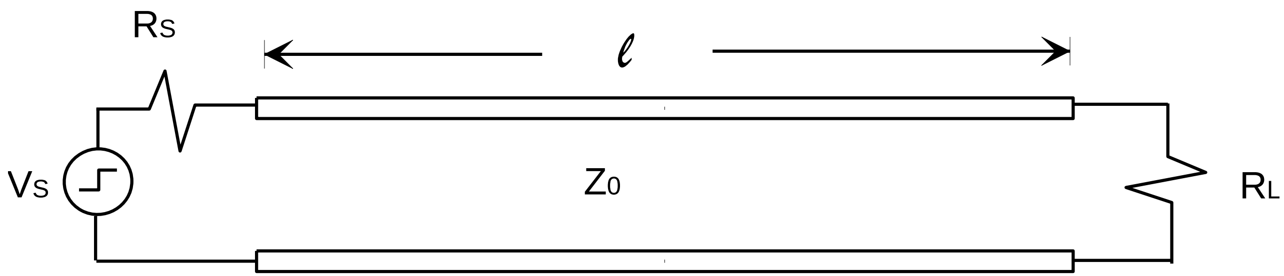


The answer depends on the application, but generally the following guidelines apply.

For digital signals: When $t_r < 2 * t_{pd}$

For RF signals: When $l > \lambda/8$

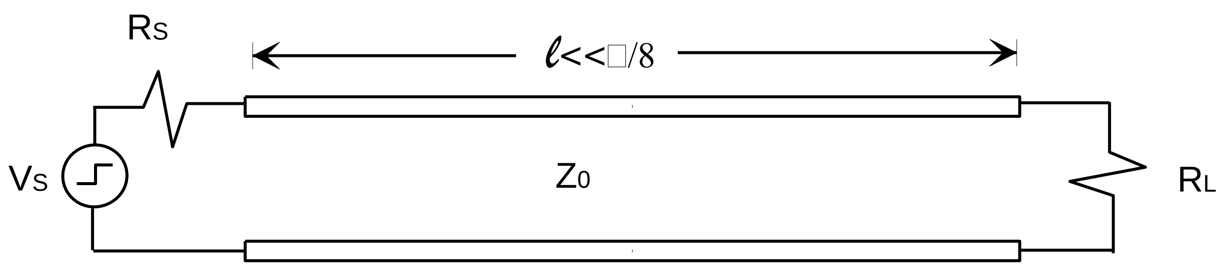
An Important Point

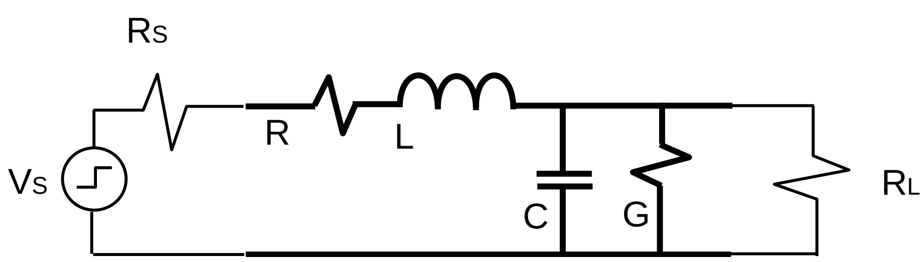


In most applications, anything that must be modeled as a transmission line must have a matched termination. This is usually undesirable from a cost and EMC perspective. Therefore, every effort should usually be taken to ensure that the signal bandwidth is no higher (or transition times are no shorter) than necessary.

When Cables are Electrically Short …

They can be modeled using their lumped RLCG parameters.

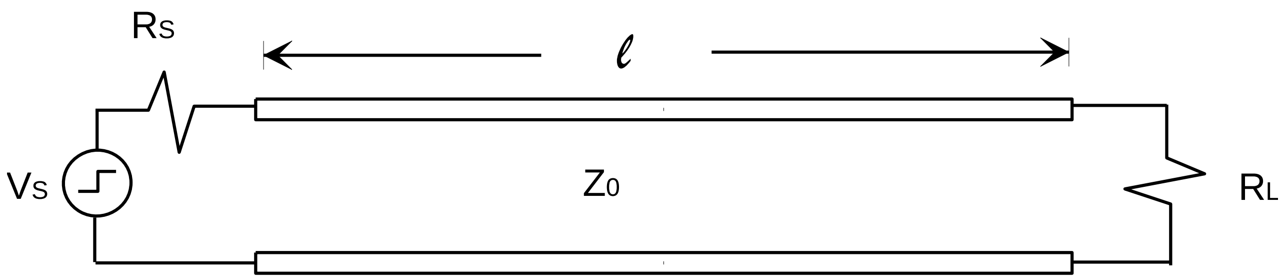




Often, one or none of these parameters is significant relative to the source and load impedances

When Cables are Not Electrically Short …

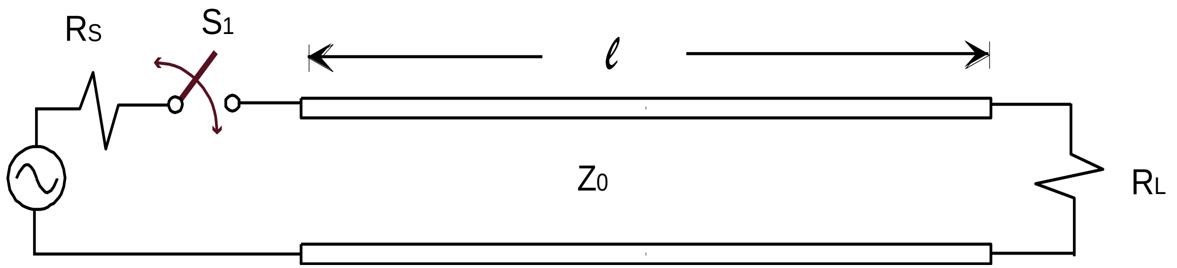
To eliminate reflections, transmission lines must have a controlled impedance and must be matched!



For signals with one source and one load, the match can occur at the source end: $\mathbf{R_S = Z_0}$.

For signals with one source and more than one load, the match must generally occur at the load end: $\mathbf{R_L = Z_0}$.

When is a Wiring Harness a Transmission Line?

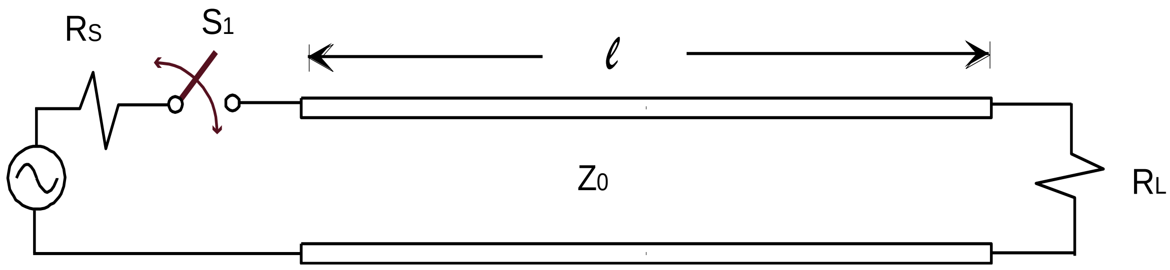


Steady state solution is always the wire-pair solution

If we don't care about how we get to the steady state, then we don't need to worry about transmission line solutions.

In most automotive applications, we don't care!

When is a Wiring Harness a Transmission Line?



If the risetime is much greater than the propagation delay, transmission line can be modeled as lumped element.

length = 5 meters → propagation delay ~ 30 nsec

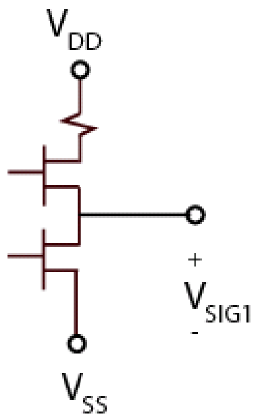
length = 50 cm → propagation delay ~ 3 nsec

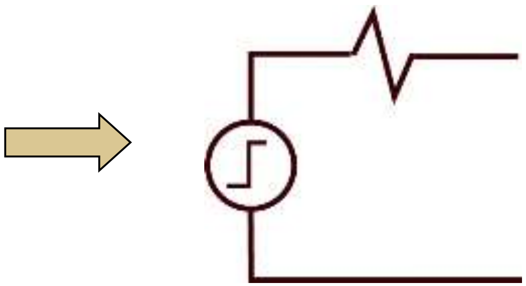
When is a Wiring Harness a Transmission Line?

- Every digital signal transition time should be forced to be >100 ns (unless this would prevent the circuit from working).
- Signals that must transition faster than 100 ns, should transition in the longest permissible time.
- Traces or cables that carry signals with transition times > 100 ns should not have matched terminations unless the length of the signal propagation is > 5 meters.
- Traces or cables that carry signals with transition times > 10 ns should not have matched terminations unless the length of the signal propagation is > 50 cm.

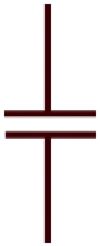
Control Transition Times

CMOS Driver Model

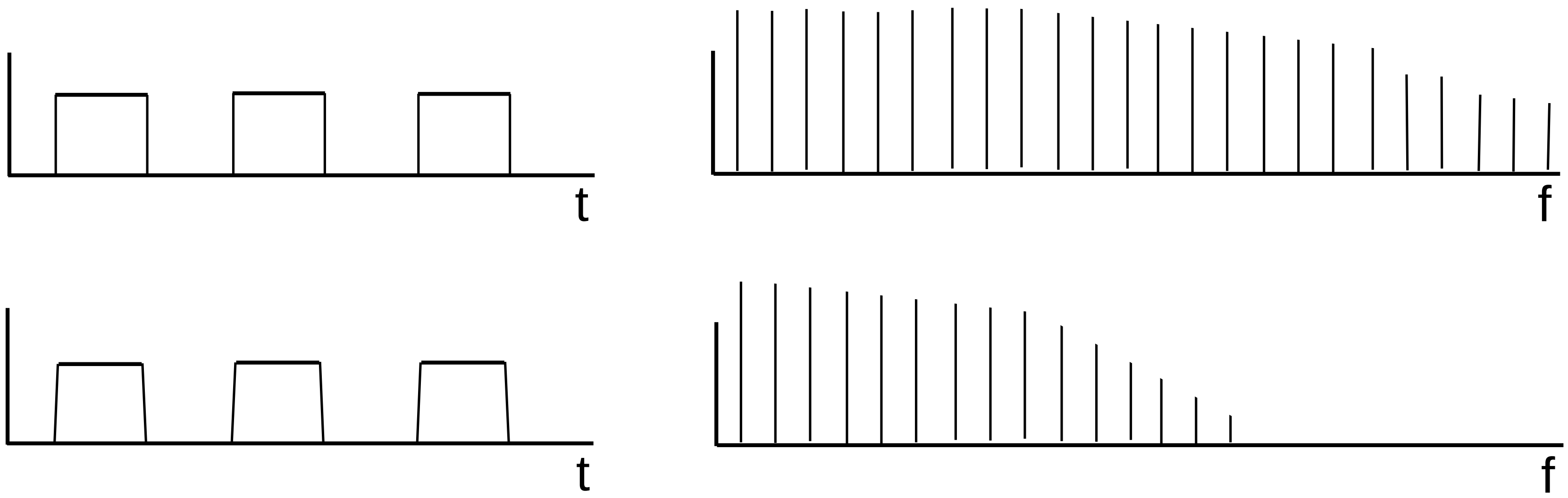




CMOS Input Model

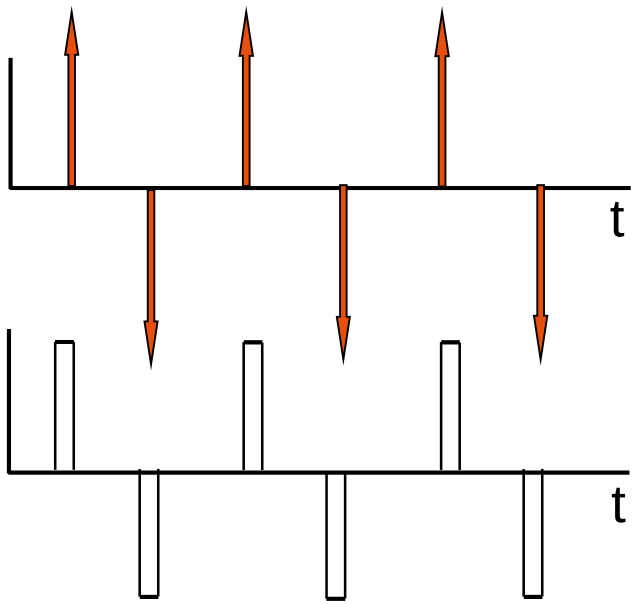


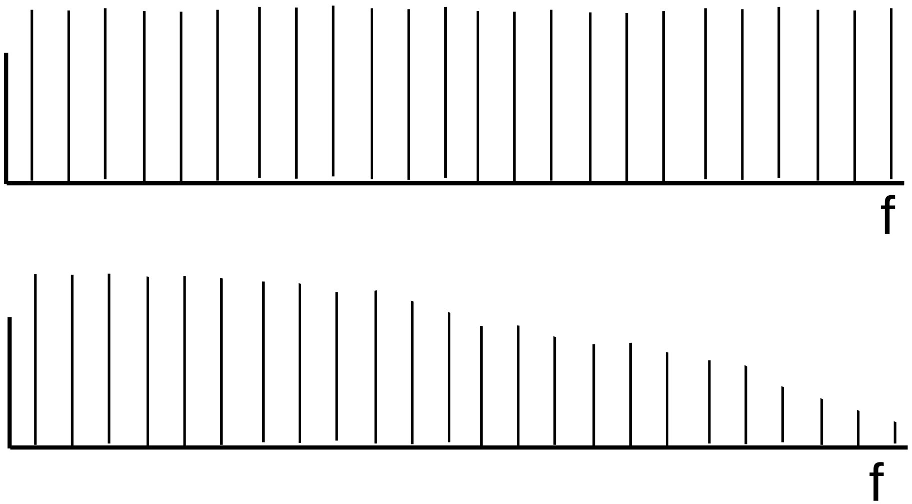
Control Transition Times



Control transition times of digital signals!

Control Transition Times





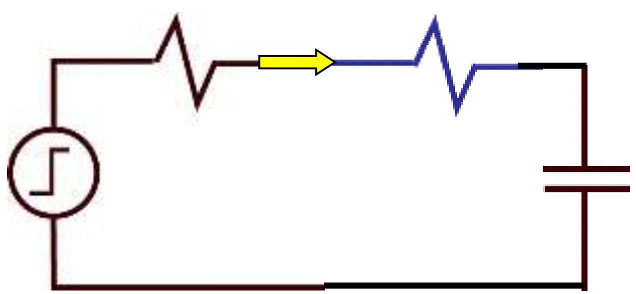
Control transition times of digital signals!

Can use a series resistor or ferrite when load is capacitive.

Use appropriate logic for fast signals with matched loads.

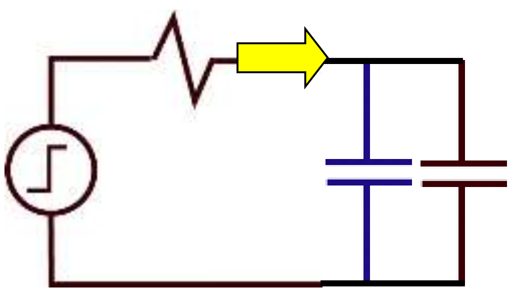
Control Transition Times

Reducing risetime with a series resistor



Good idea

Reducing risetime with a parallel capacitor



Bad idea

Example 1: Microcontroller Output Driver

Automotive microcontroller in typical application:

Suppose we connected an output of this microcontroller directly up to an impedance-matched antenna…

Available Information

V_{source} = 3.3 V

I_{max} = 20 mA

C_{in} = 5 pF

R_{series} = 0 Ω

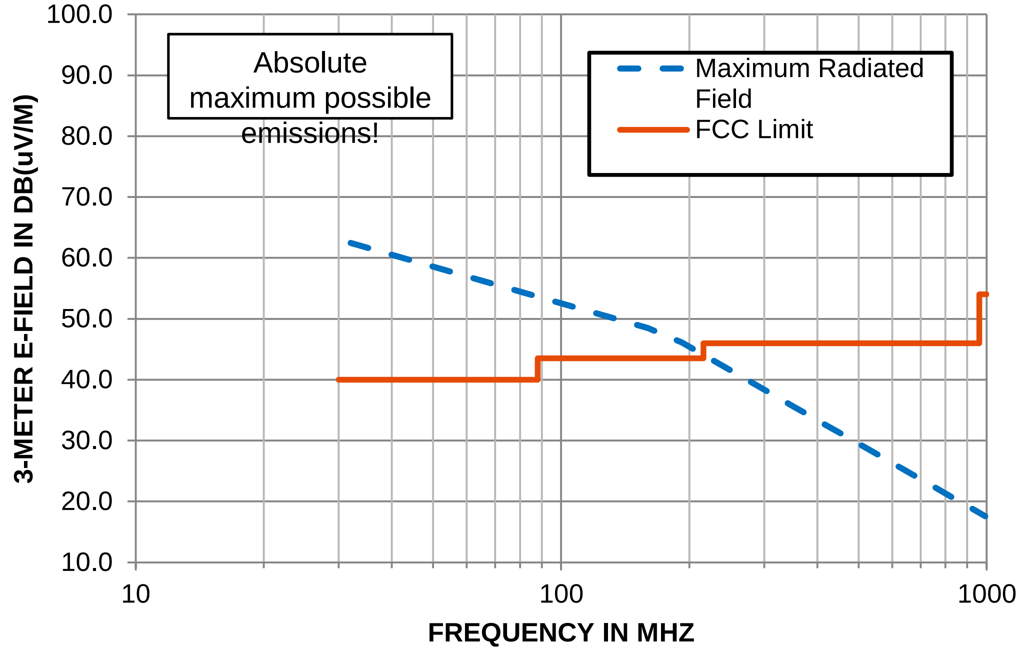
CLK Freq = 100 kHz

Calculated Parameters

R_{source} = 165 Ω

T = 10 μs

t_r = 1.82 ns



Example 1: Microcontroller Output Driver

Same output with 20-kΩ series resistor:

Suppose we connected an output of this microcontroller directly up to an impedance-matched antenna…

Available Information

V_{source} = 3.3 V

I_{max} = 20 mA

C_{in} = 5 pF

R_{series} = 20 kΩ

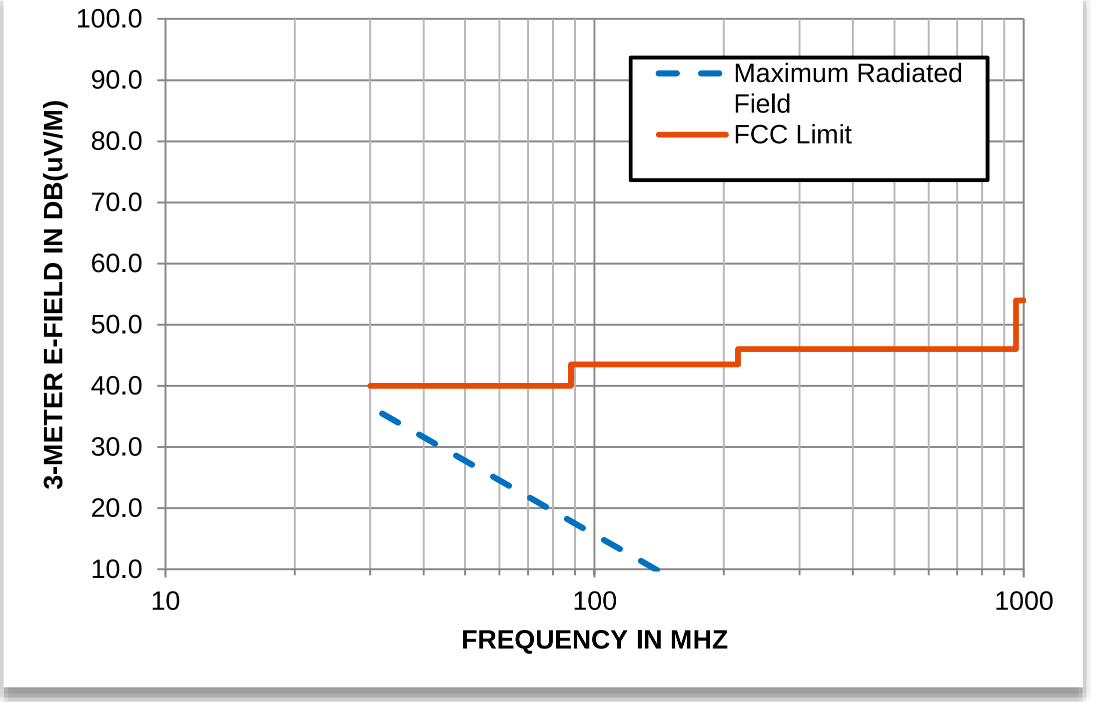
CLK Freq = 100 kHz

Calculated Parameters

R_{source} = 8165 Ω

T = 10 µs

t_r = 220.0 ns



Example 2: Microcontroller Output Driver

Same output with 1 MHz output:

Suppose we connected an output of this microcontroller directly up to an impedance-matched antenna…

Available Information

V_{source} = 3.3 V

I_{max} = 20 mA

C_{in} = 5 pFa

R_{series} = 0 kΩ

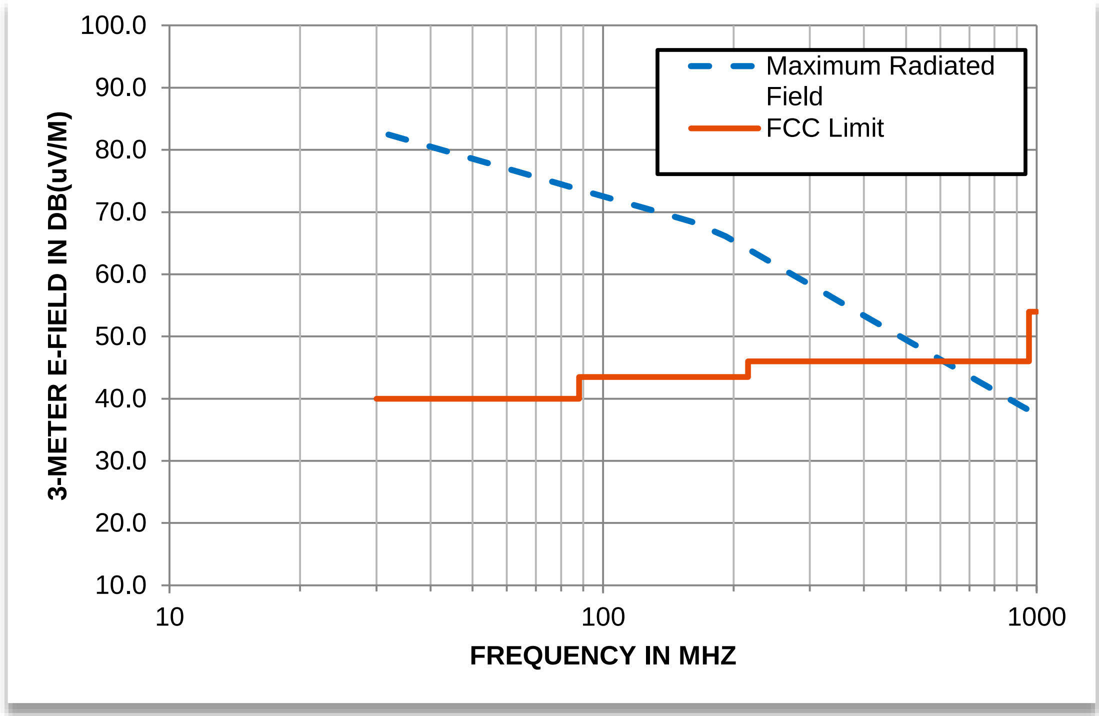
CLK Freq = 1 MHz

Calculated Parameters

R_{source} = 165 Ω

T = 1 μs

t_r = 1.82 nsaaa



Example 2: Microcontroller Output Driver

Same output with 1 MHz output and 8-kΩ series resistor:

Suppose we connected an output of this microcontroller directly up to an impedance-matched antenna…

Available Information

V_{source} = 3.3 V

I_{max} = 20 mA

C_{in} = 5 pF

R_{series} = 8 kΩ

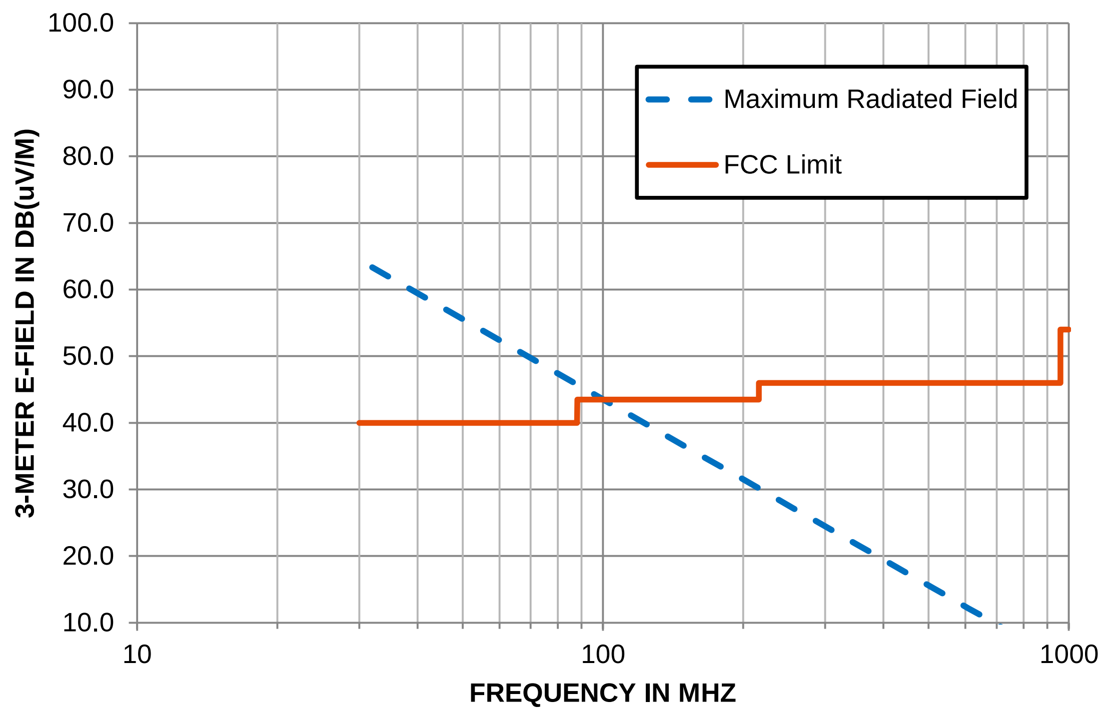
CLK Freq = 1 MHz

Calculated Parameters

R_{source} = 8165 Ω

T = 1 μs

t_r = 90 ns

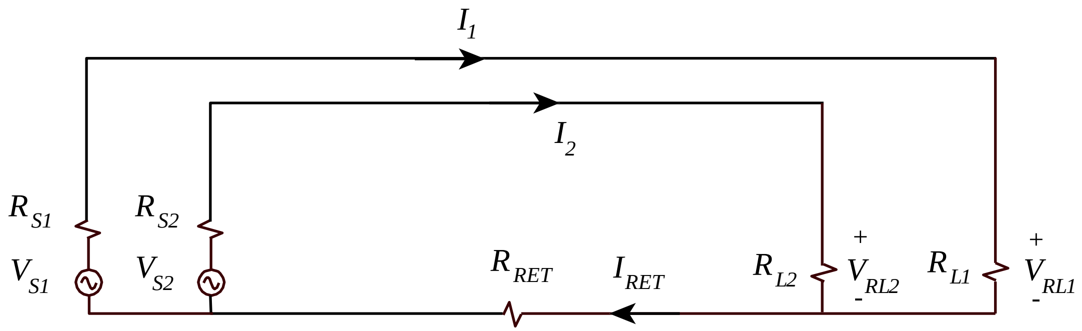


Automotive and Industrial Design for EMC

To guarantee that your design will meet its EMC requirements the first time, you **must**:

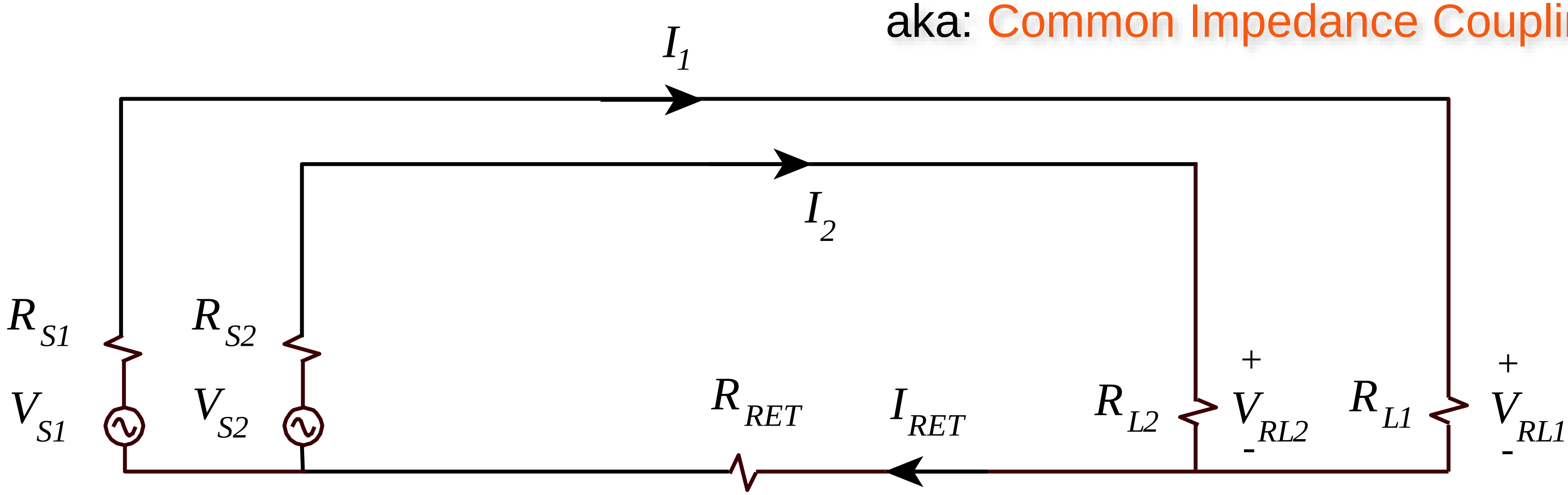
Recognize the four (not two) possible coupling mechanisms!

Common Impedance Coupling



Conducted Coupling

aka: Common Impedance Coupling



Requires 2 conductor connections between the source and victim.

The only mechanism that couples DC level shifts.

Most likely to be dominant at low frequencies, when source and victim share a current return path.

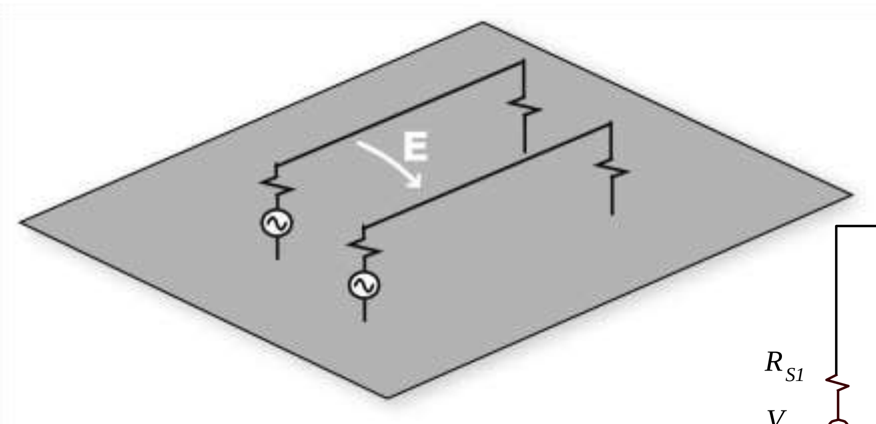
Most likely to be dominant when sources are low impedance (high current) circuits.

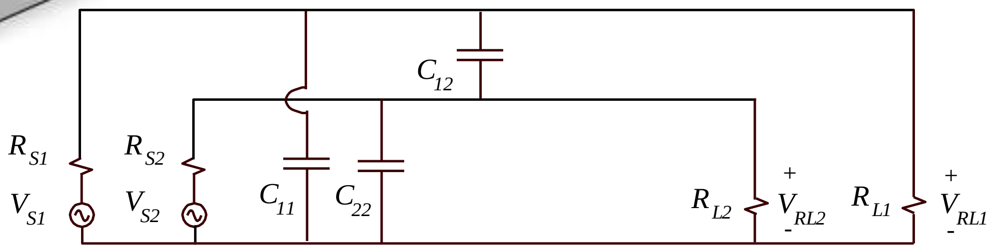
Conducted Coupling Examples

- Lights dim and radio dies when automobile engine is started.
- Power bus voltage spikes are heard as audible “clicks” on an AM radio using the same power source.
- An electrostatic discharge transient resets a microprocessor causing a system to shutdown.
- A lightning induced transient destroys the electronic components in a computer with a wired connection to the internet.

Electric Field Coupling

aka: Capacitive Coupling





Requires 0 conductor connections between the source and victim.

Coupling proportional to dV/dt.

Most likely to be dominant at higher frequencies.

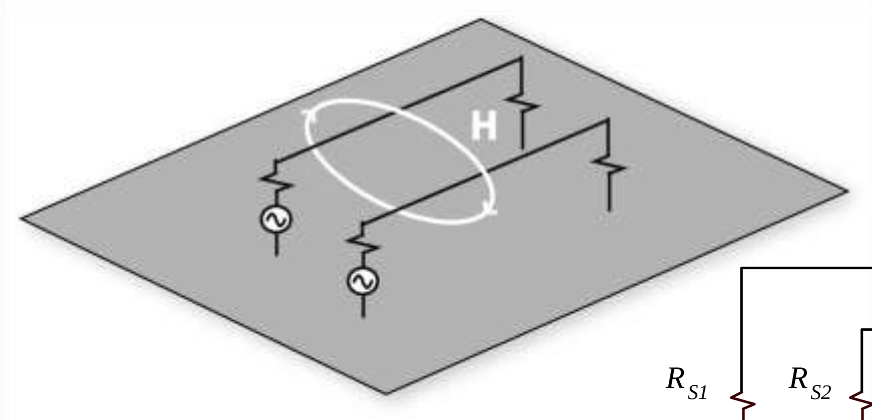
Most likely to be dominant when sources are high impedance (high voltage) circuits.

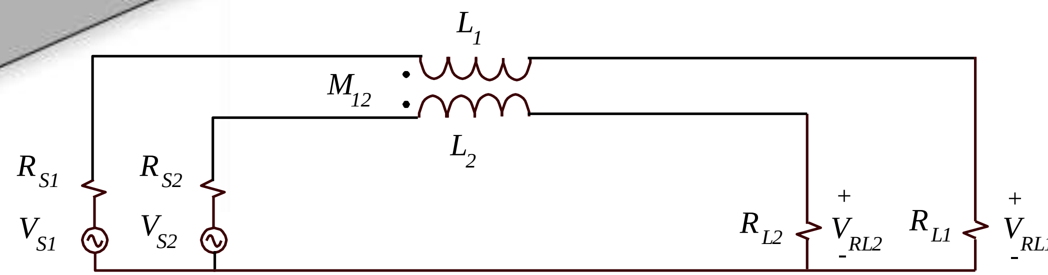
Electric Field Coupling Examples

- Coupling from circuit board heatsinks to cables or enclosures.
- AM radio interference from overhead power lines.
- Automotive component noise picked up by the rod antenna in CISPR 25 “radiated” emissions tests.
- Microprocessor resets due to indirect electrostatic discharges.

Magnetic Field Coupling

aka: Inductive Coupling





Requires 0 conductor connections between the source and victim.

Coupling proportional to dI/dt.

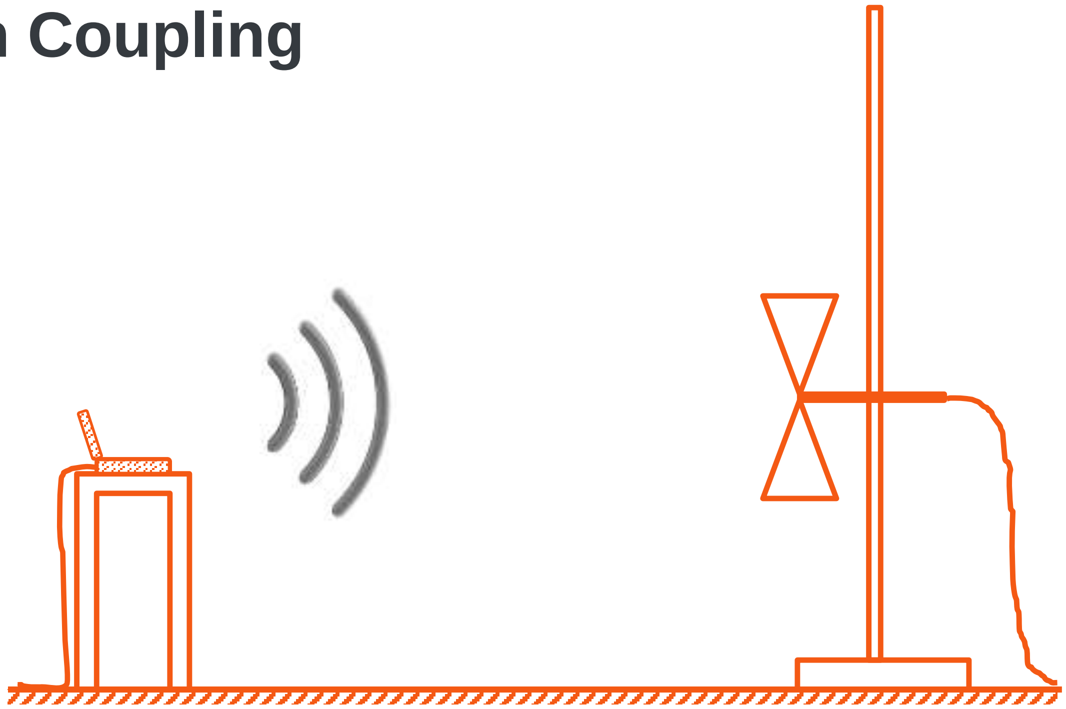
Most likely to be dominant at higher frequencies.

Most likely to be dominant when sources are low impedance (high current) circuits.

Magnetic Field Coupling Examples

- Coupling from power transformers or fluorescent lighting ballasts.
- Jitter in CRT displays.
- 60 Hz “hum” in a handheld AM radio.
- Hard-drive corruption due to motor or transformer currents.

Radiation Coupling



Requires 0 conductor connections between the source and victim.

Requires at least a wavelength of separation between source and victim.

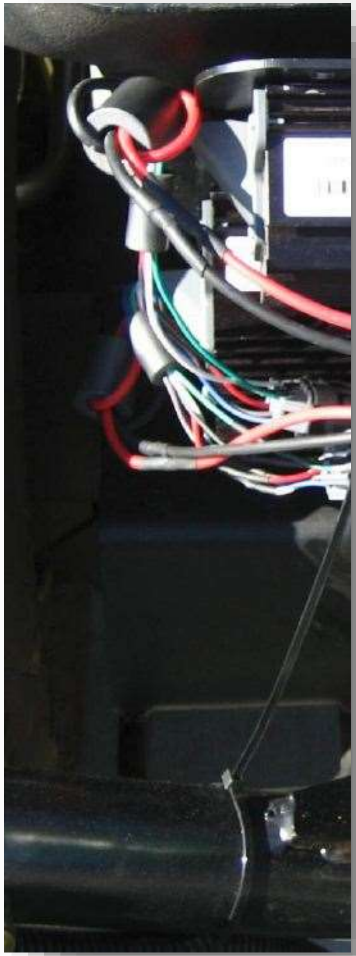
Requires a transmitting “antenna” and a receiving “antenna”.

Radiation Coupling

- To guarantee that your design will meet its radiated emissions and immunity requirements the first time, you **must**:

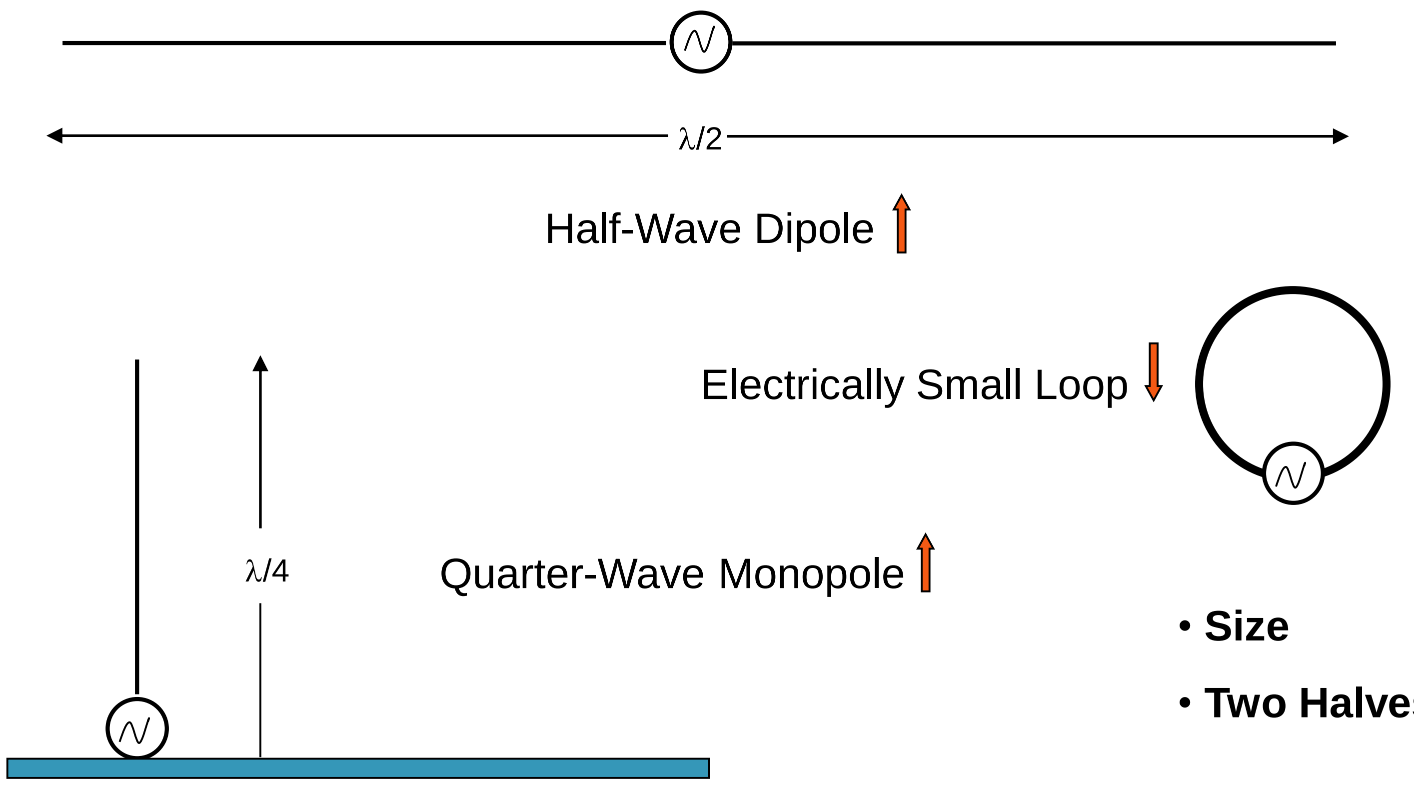
Identify your antennas!

When Do Wiring Harnesses Look Like Antennas?



Identifying Antennas

What makes an efficient antenna?



- **Size**
- **Two Halves**

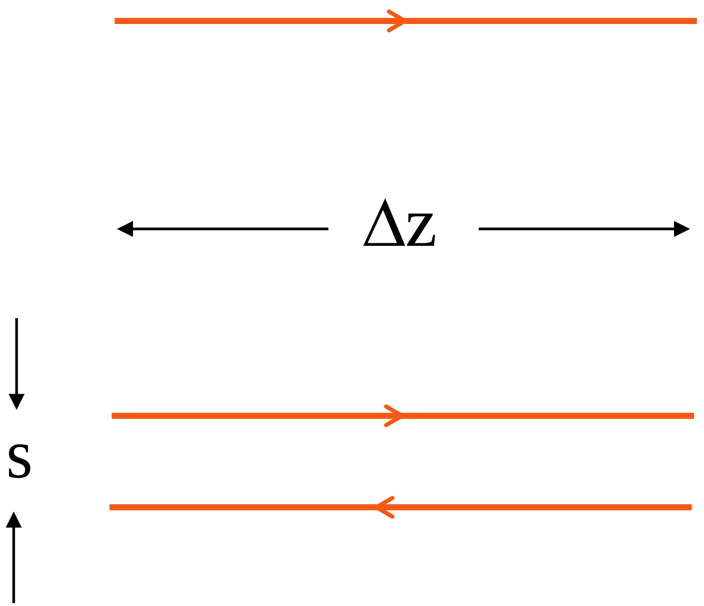
Identifying Antennas

Good Antenna Parts		Poor Antenna Parts	
<100 MHz	**>100 MHz**	**<100 MHz**	**>100 MHz**
Cables	Heatsinks	Microstrip or stripline traces	Microstrip or stripline traces
	Power planes		
	Tall components	Anything that is not big	
	Seams in shielding enclosures		

Free-space wavelength at 100 MHz is 3 meters

Identifying Antennas

Common-Mode vs. Differential Mode



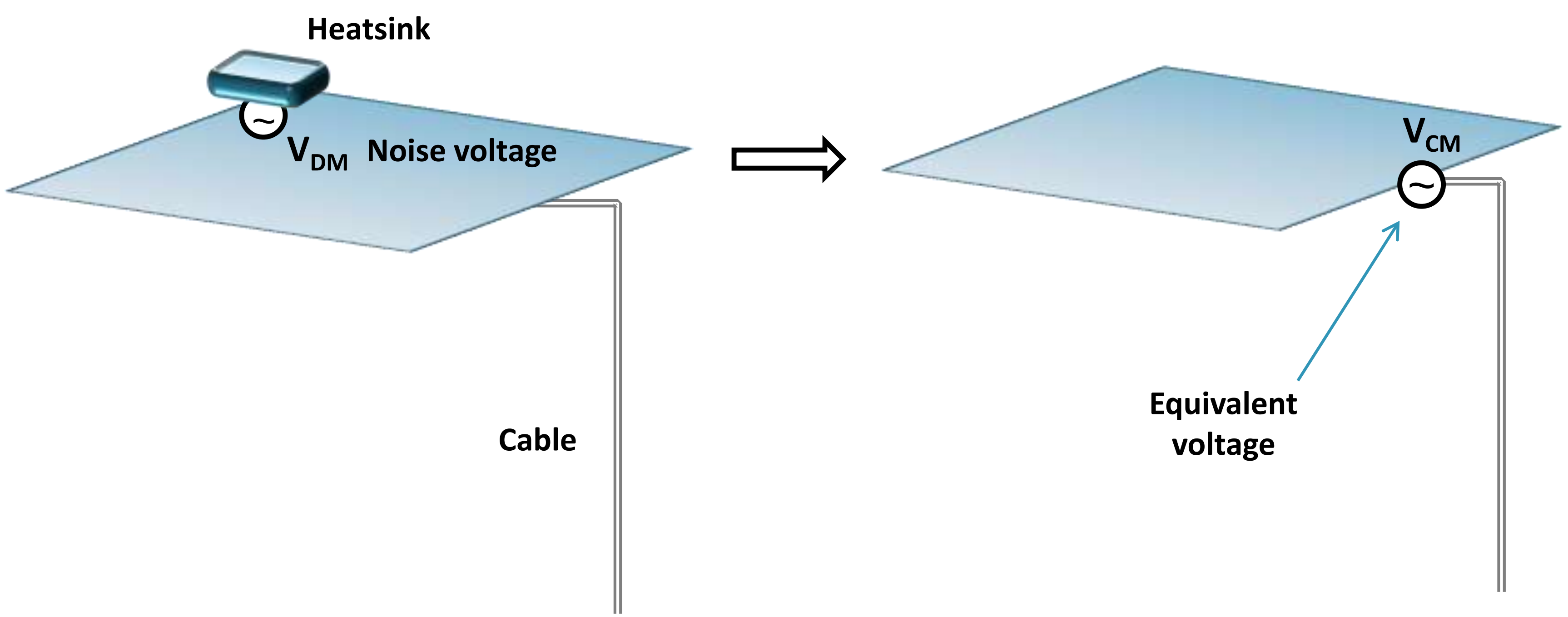
$$E_{max} = 1.26 \times 10^{-6} \frac{|I_c| f \Delta z}{r}$$

$$E_{max} = 1.32 \times 10^{-14} \frac{|I_d| f^2 s \Delta z}{r}$$

$$= 4 \times 10^{-6} \frac{|I_d| f \Delta z}{r} \left(\frac{s}{\lambda} \right)$$

How are Common-mode Currents Induced on Cables?

Voltage-Driven Mechanism



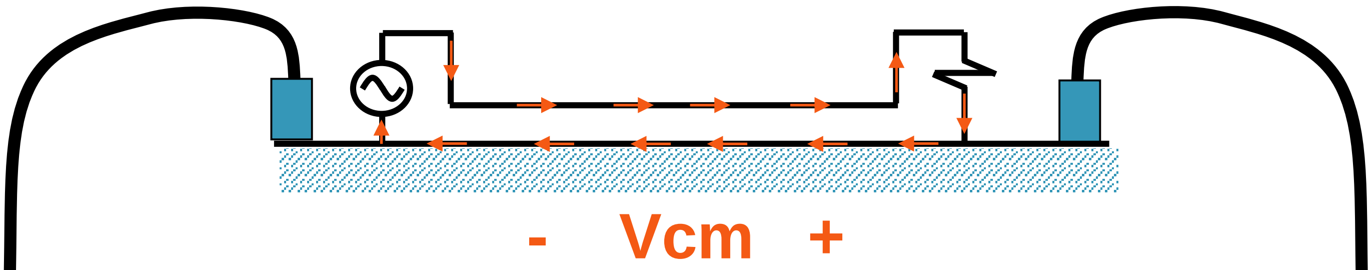
$$V_{CM} = \frac{C_{heat\sin k}}{C_{board}} V_{DM}$$

$$V_{DM} = 0.2234 \frac{C_{board}}{C_{heat\sin k}} \times \frac{r}{F_{board} F_{cable}} \times |E_{\max}|$$

How are Common-mode Currents Induced on Cables?

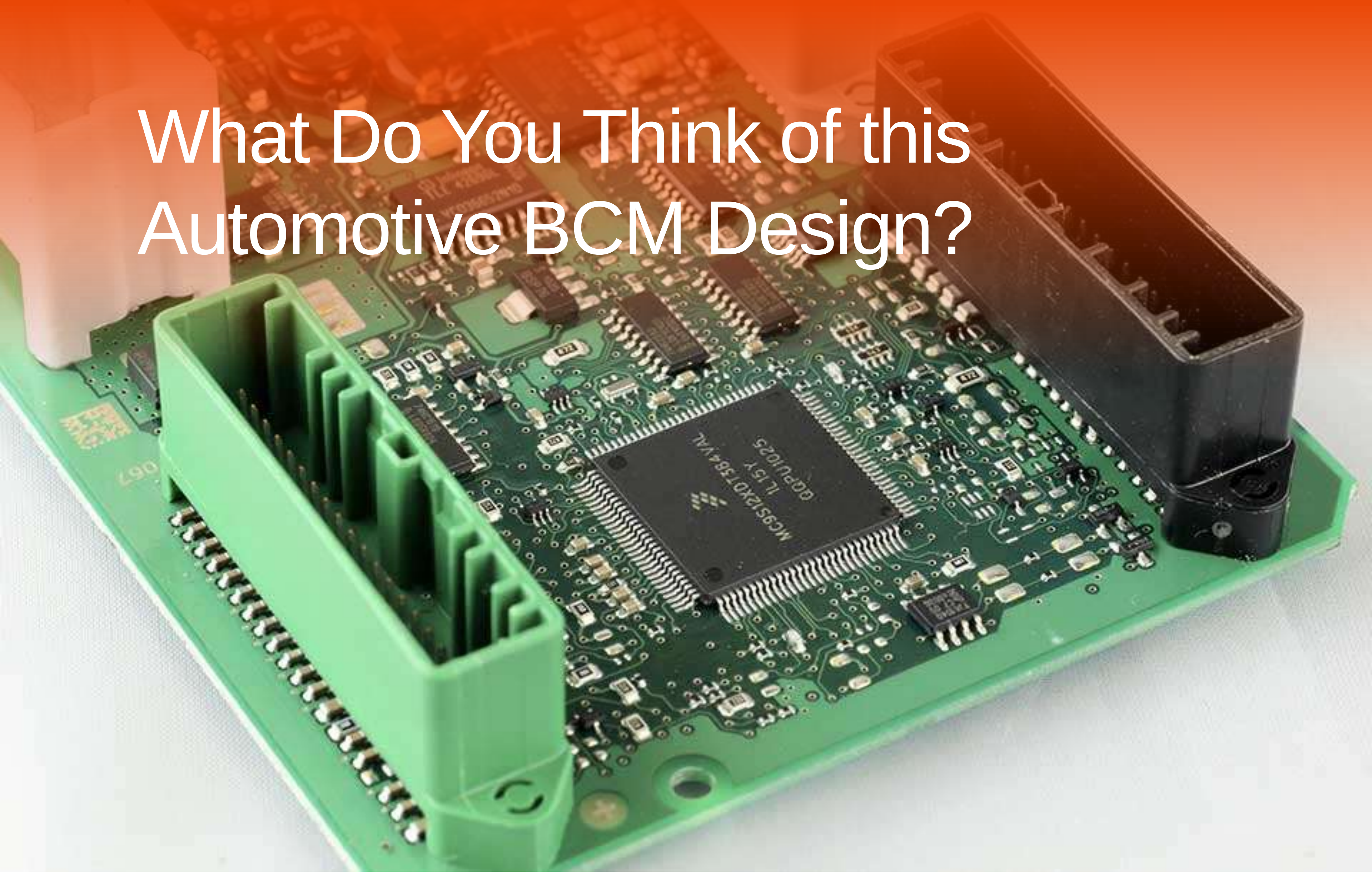
Current-Driven Mechanism

Signal current loop induces a voltage between two good antenna parts.



At 10 MHz and higher, milliamps of current flowing in a ground plane produces millivolts of voltage across the ground plane. A few millivolts driving a resonant antenna can result in radiated fields exceeding FCC limits.

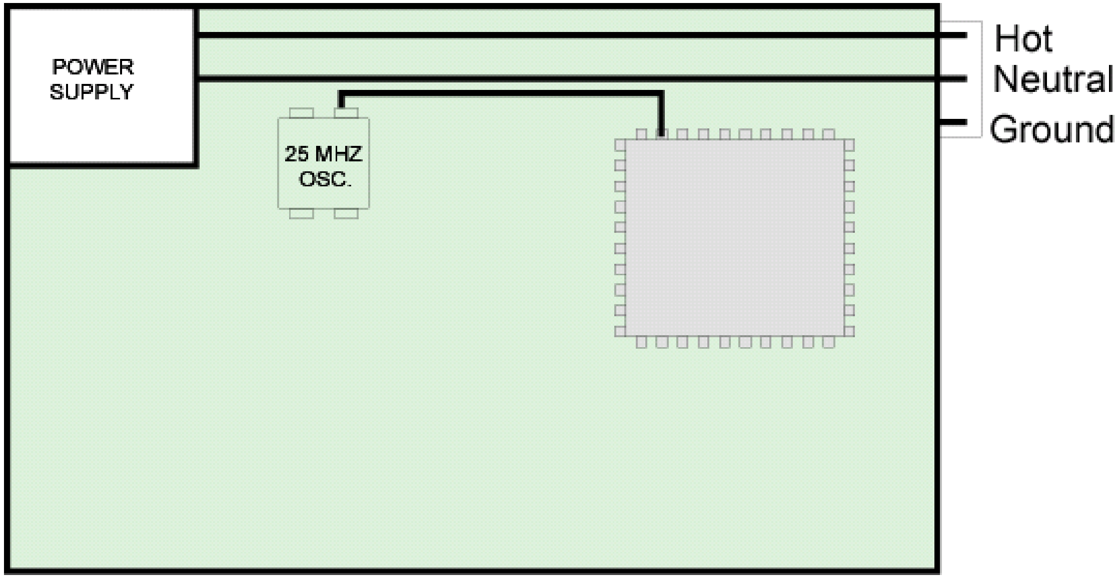
What Do You Think of this Automotive BCM Design?



How are Common-mode Currents Induced on Cables?

Direct-Coupling Mechanism

Signals coupled to I/O lines can carry HF power off the board.

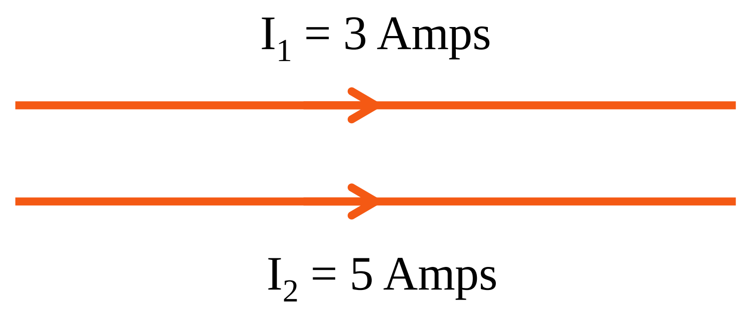


Common-mode and Differential-mode Current

General Definition

$$I_{DM} = (1-h)I_1 - hI_2$$

$$I_{CM} = I_1 + I_2$$



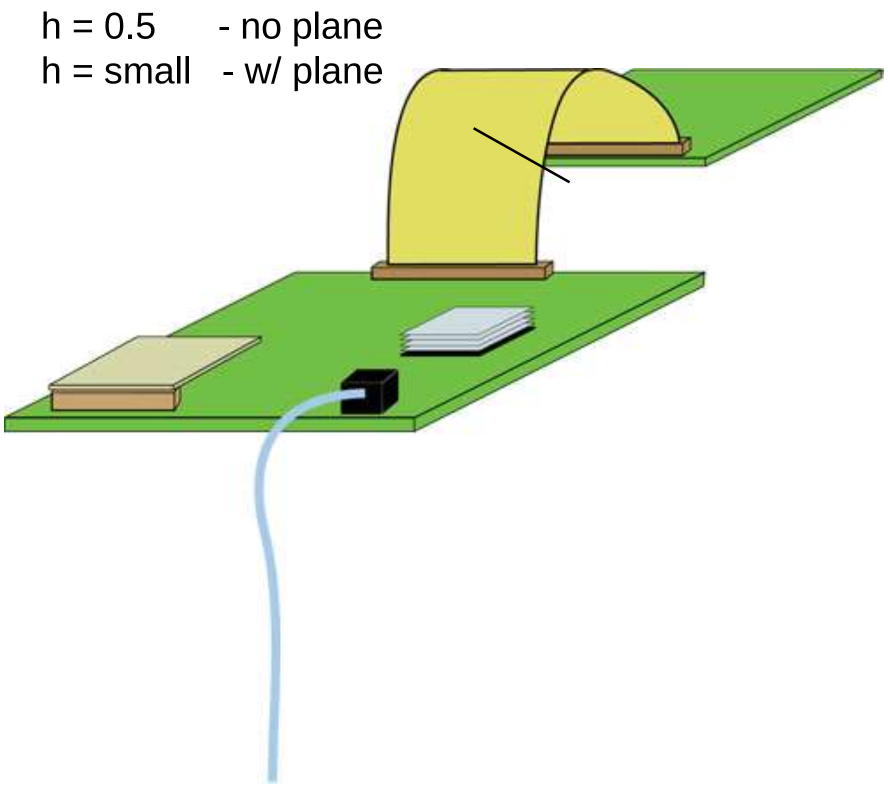
$$I_1 = I_{DM} + hI_{CM}$$

$$I_2 = -I_{DM} + (1-h)I_{CM}$$

<u>AND</u>

$$I_{DM} = \frac{V_{DM}}{Z_{DM}}$$

Driving a Ribbon Cable

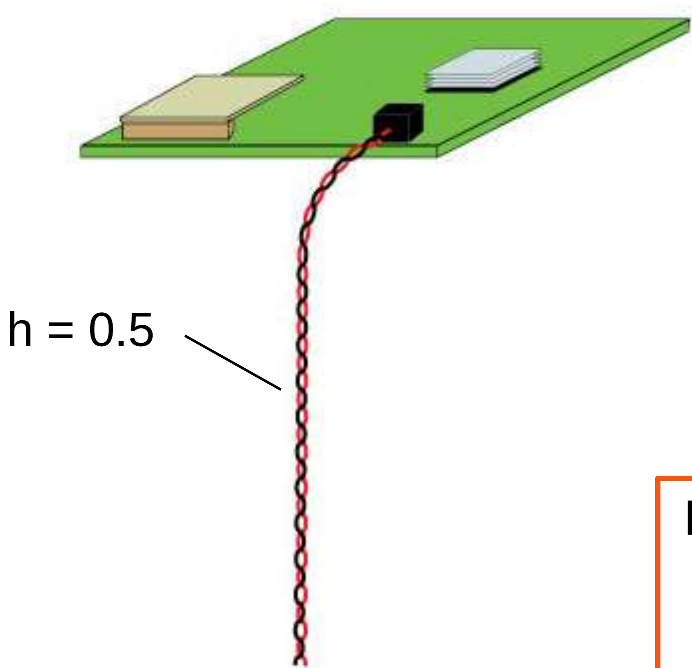


A perfect differential driver driving two adjacent wires in a ribbon cable produces no common-mode current on the ribbon cable.

A single-ended driver driving two adjacent wires in a ribbon cable produces a exactly the same amount of common-mode current as a common-mode source with half the signal voltage

Don't drive ribbon cable wires with single-ended sources unless you know the common-mode current will not be a problem.

PCB Driving a Twisted Wire Pair



A perfect differential driver driving a perfect twisted-wire pair produces no common-mode current on the wire pair.

A single-ended driver driving a twisted-wire pair produces a exactly the same amount of common-mode current as a common-mode source with half the signal voltage

Don't drive twisted-wire pairs with single-ended sources unless you know the common-mode current will not be a problem.

Automotive and Industrial Design for EMC

- To guarantee that your design will meet its EMC requirements the first time, you must:

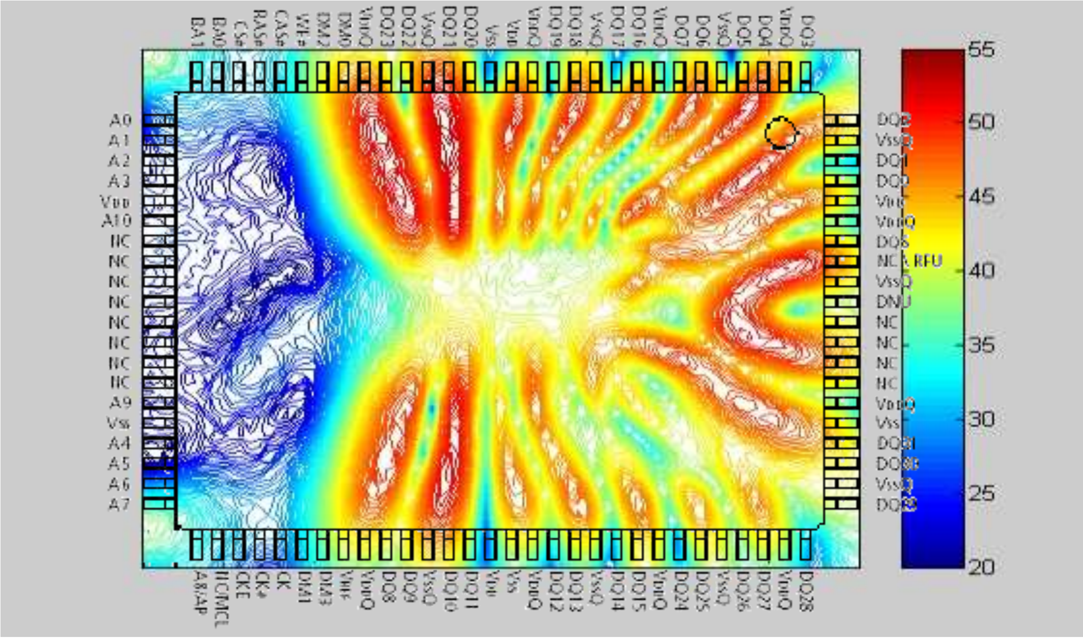
Identify your sources!

Identify Sources

Clocks	Narrow band, consistent
Digital Data	Not as narrow as clocks, but clock frequency is usually identifiable.
Analog Signals	Bandwidth determined by signal source, consistent
Power Supply Switching	Appears broadband, but harmonics of switching frequency can be identified, consistent
Arcing	Broadband, intermittent
Parasitic Oscillations	Narrowband, possibly intermittent

Identify Sources

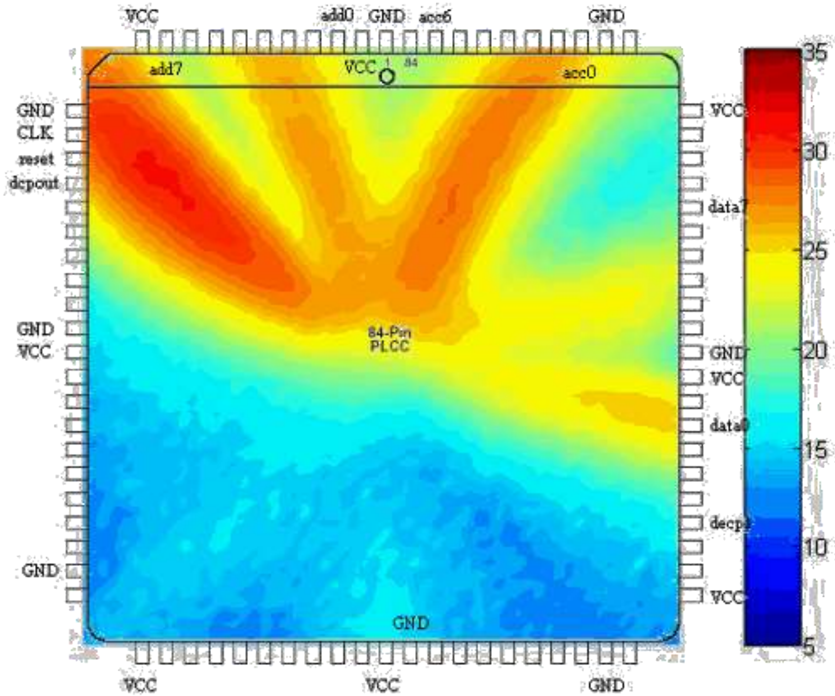
Active Devices (Power Pins)



For some ICs, the high-frequency currents drawn from the power pins can be much greater than the high-frequency currents in the signals!

Identify Sources

Noise on the low-speed I/O



For some ICs, significant high-frequency currents appear on low-speed I/O including outputs that never change state during normal operation!

Automotive and Industrial Design for EMC

- To guarantee that your design will meet its EMC requirements the first time:

Don’t rely on EMC design guidelines!

EE371 Design Problem

UNIVERSITY OF MISSOURI-ROLLA
ELECTROMAGNETIC COMPATIBILITY LABORATORY

Printed Circuit Board Layout Guidelines

1. Keep loop areas small (signal paths, decoupling) (minimize inductance, radiation).
2. Place components before routing traces.
3. Locate components to minimize length of high frequency and I/O lines.
4. Route ground first, then power, then I/O traces, then high-speed traces, then slower traces.
5. Connectors are the best interface to possible antennas! Don't allow high speed signals to flow between connectors. Place all I/O on one edge of the board if possible.
6. Avoid letting I/O come too far on to the board and keep I/O away from high frequency lines.
7. Leave space for a filter or choke on I/O lines during layout.
8. Unused logic gate inputs should be tied to ground or Vcc.
9. Spacing between any trace and the board edge should be greater than the height of the trace above its return plane.
10. Traces on adjacent layers should be at right angles to each other.
11. Do not permit floating areas of metal.
12. Fill empty areas of the board with metal grounded to the return plane.
13. If possible, the highest speed traces should be sandwiched between planes.
14. Do not allow pigtail connections of ground to connector shell. Use low inductance connections.
15. Lateral separation is more effective than vertical separation.
16. Choose logic families that are no faster than necessary.

Power and Ground Distribution

1. Provide low inductance power and ground to every active component.
2. Layout the ground first, then power, then I/O, then the high frequency lines, then the rest.
3. Divide circuit board into different DC power voltage areas.
4. Do not allow different DC voltage planes to overlap one another.
5. Gaps can be used for kHz currents, but when the ground plane is gapped, all planes should be gapped in the same place.
6. Traces must not cross a gap in the ground plane.
7. The board should have ONE well-defined ground at or near the connectors (don't split).
8. Watch out for via holes cutting slots in the ground plane.
9. Provide at least one decoupling capacitor for each I.C.
10. Provide bulk decoupling where power comes on to the board (~10X the sum of all other decoupling caps).
11. Minimize the series inductance of any lumped decoupling capacitors.

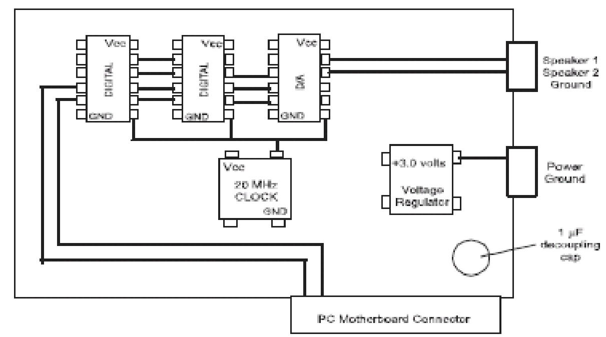
Connector Pin Assignments

1. Layout the board first, then assign connector pins.
2. Provide plenty of ground pins (all connected directly to the return plane).
3. Separate high level and low level signal pins. Use power or return pins to help provide isolation.
4. Any extra pins should be connected to ground.

EE371 Homework #9 due 11/11/03

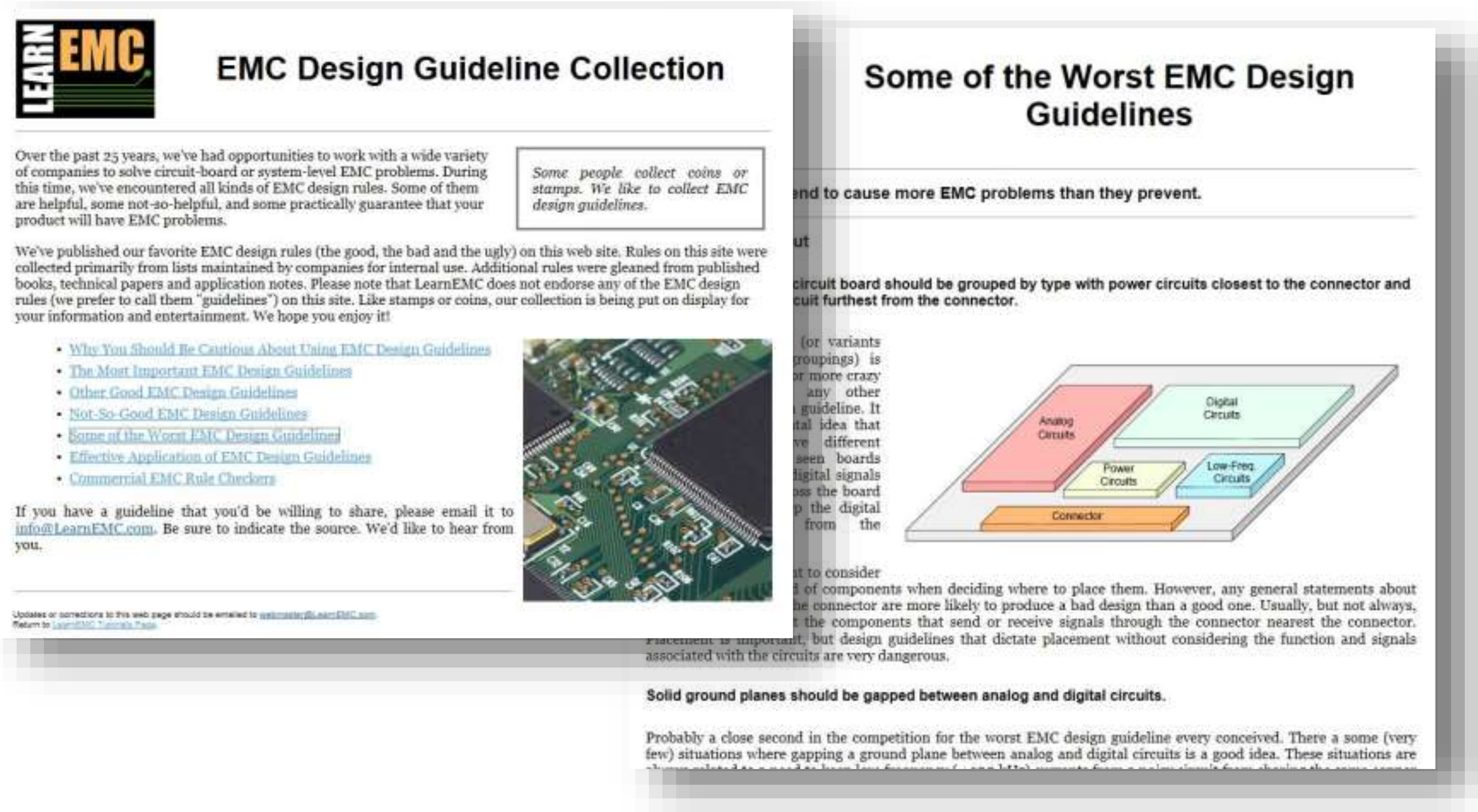
Problem 1: (30 points) A simplified design for a personal computer sound card is shown below. The board has 4 layers with the components on top. The traces shown are on layer 1. Layer 2 is a solid +3.3-volt power plane. Layer 3 is a solid ground plane. Layers 2 and 3 are spaced 40 mils (1 mm) apart. Layer 4 is empty. The board is powered by the motherboard. The board is capable of supplying +3.0-volt power to external speakers through a separate connector.

This device fails to meet the radiated emissions requirements. Show how you would layout the board to reduce radiated emissions. Don't forget to add decoupling capacitors.



EMC Design Guideline Collection

http://www.learnemc.com/tutorials/guidelines.html



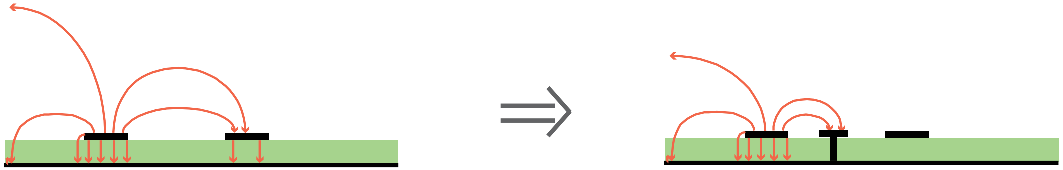
Automotive and Industrial Design for EMC

- To guarantee that your design will meet its EMC requirements the first time, you must recognize that:

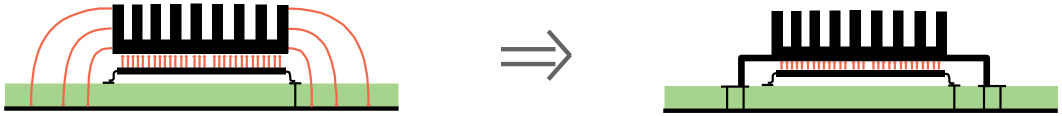
Use the right shield for the right application!

Shielding

Electric Field Shielding



(a.)



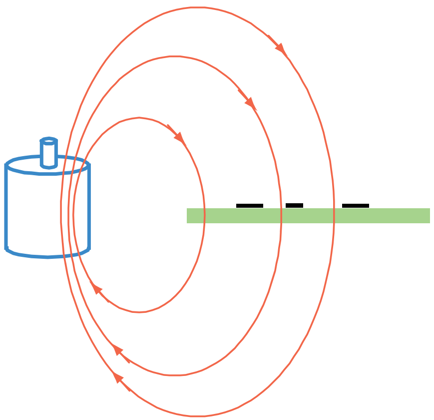
(b.)

(c.)

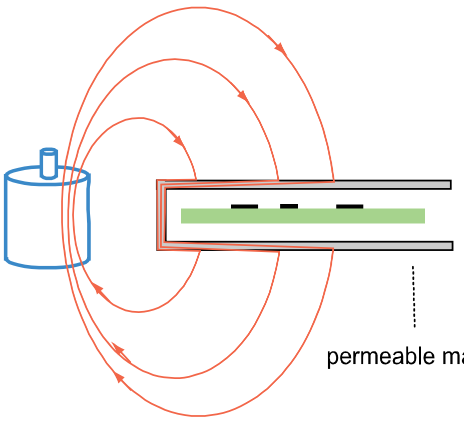
Shielding

Magnetic Field Shielding

(at low frequencies)



(a.)

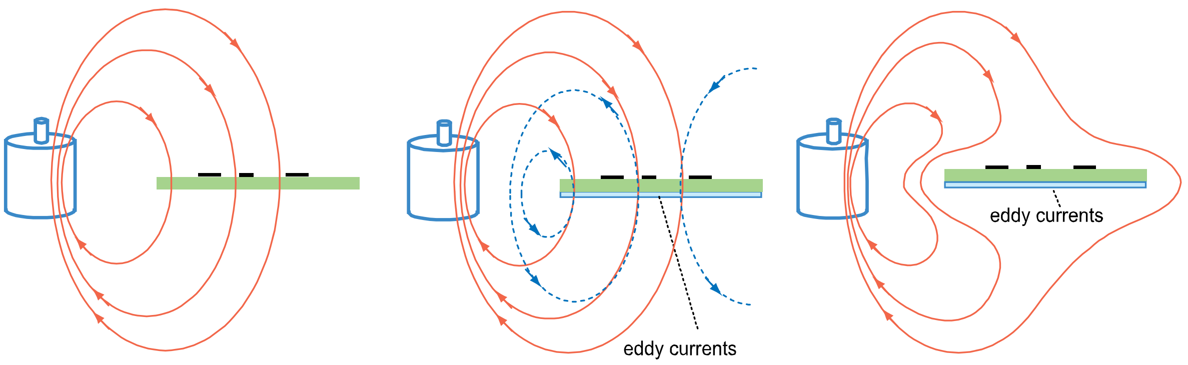


(b.)

Shielding

Magnetic Field Shielding

(at high frequencies)



(a.) (b.) (c.)

Shielding

Enclosure

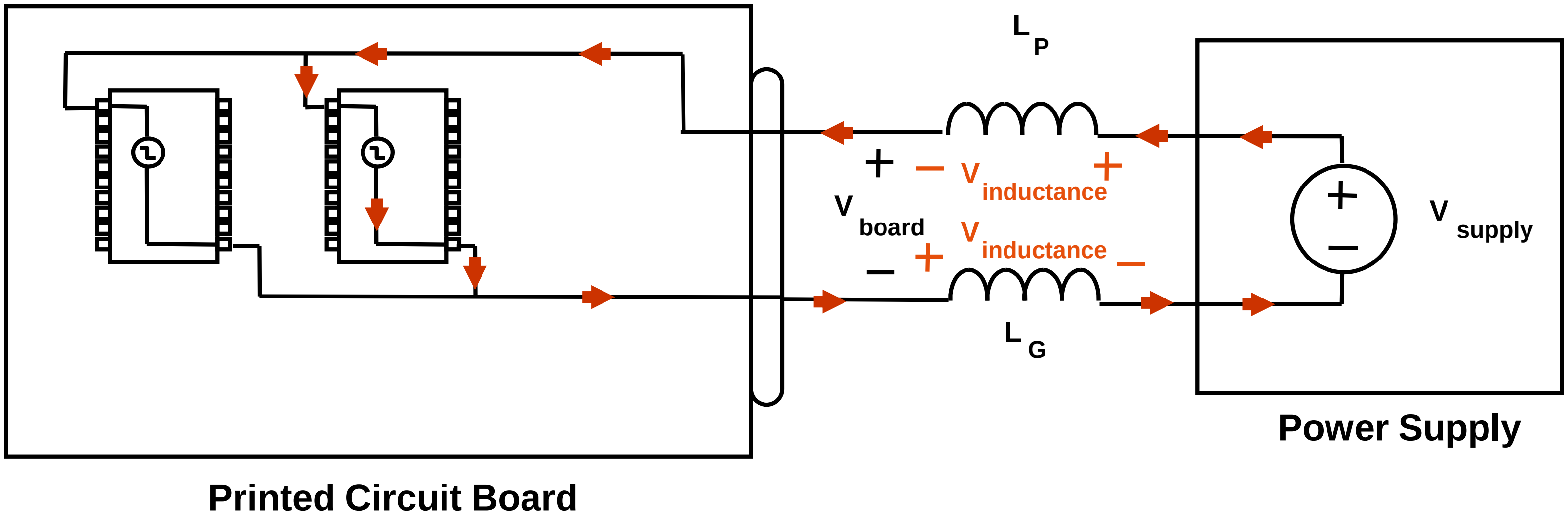
Shielding

Automotive and Industrial Design for EMC

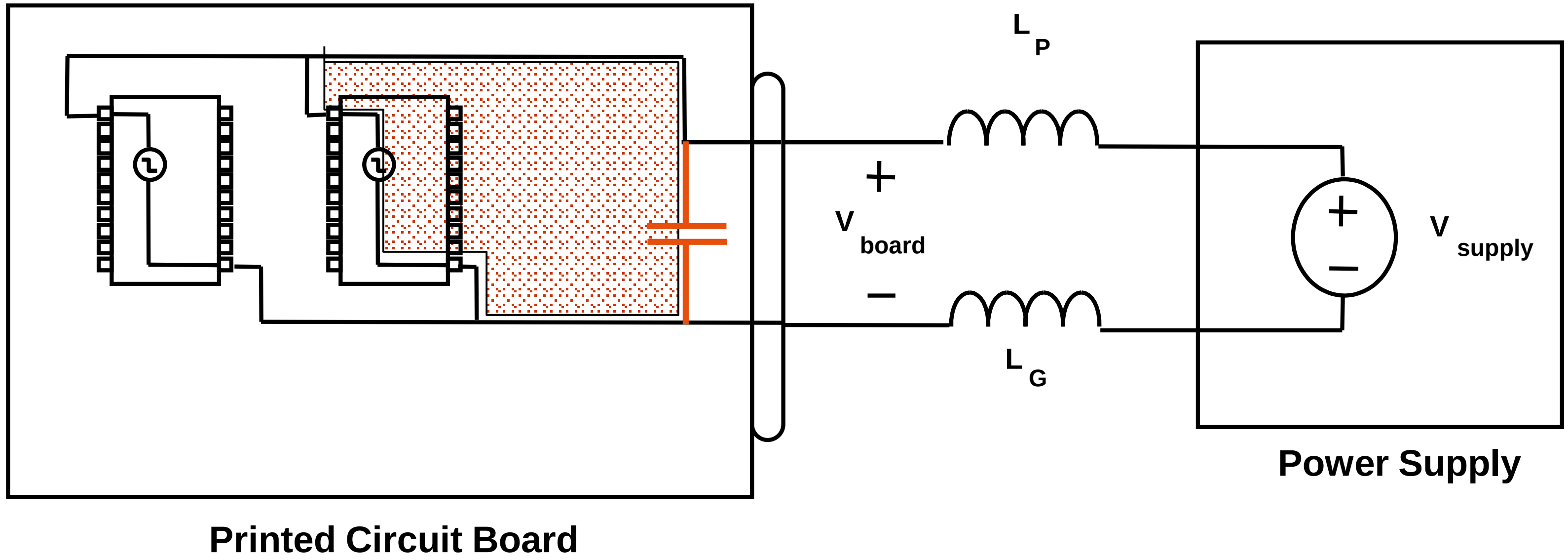
- To guarantee that your design will meet its EMC requirements the first time, you must:

Provide adequate power bus decoupling!

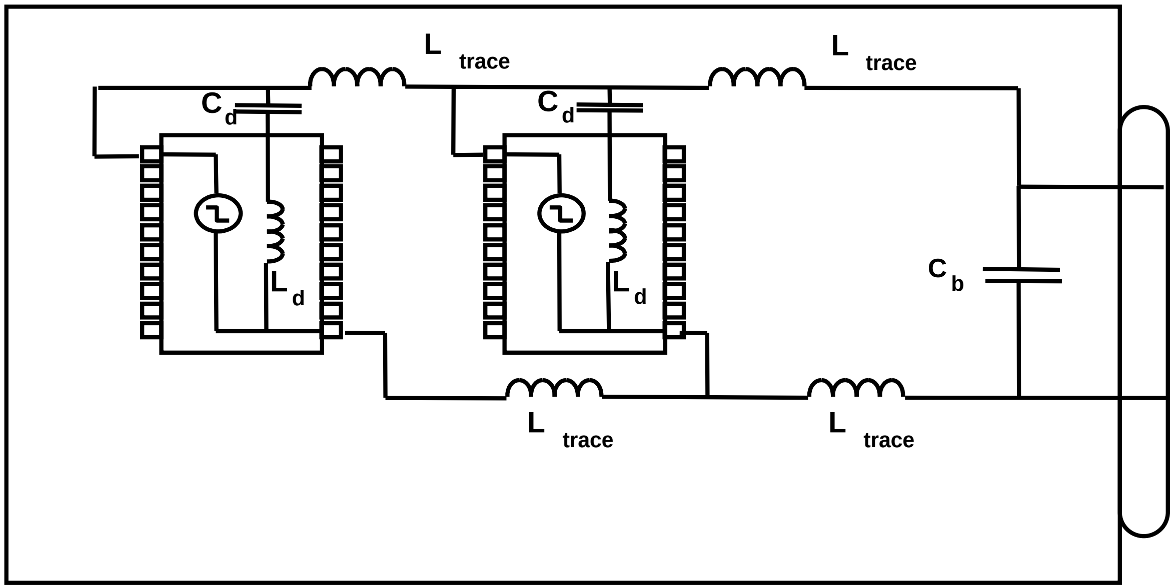
The Concept of Power Bus Decoupling



The Concept of Power Bus Decoupling



The Concept of Power Bus Decoupling



Rules for PCB Decoupling

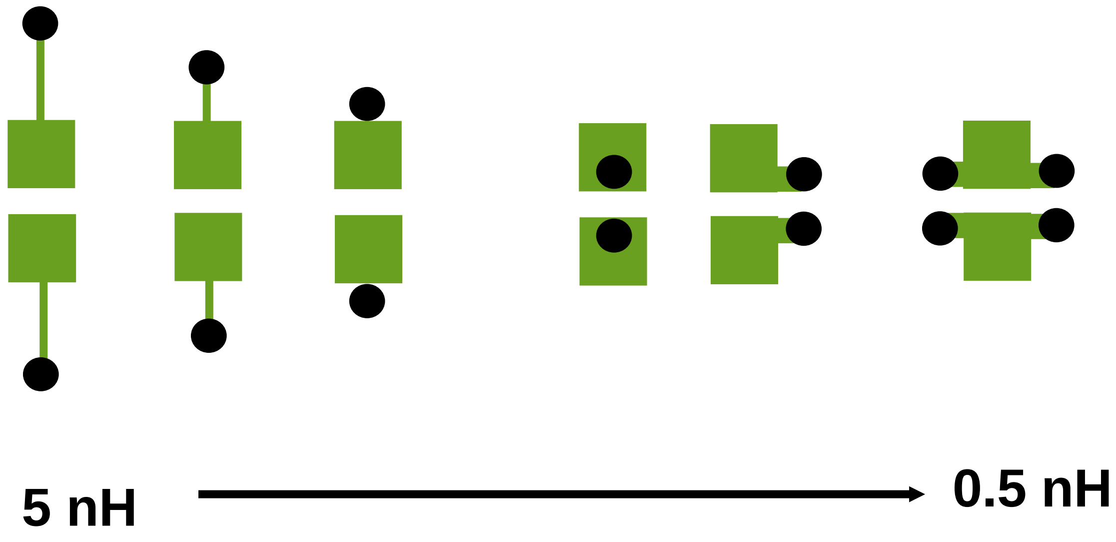
- For boards with no power or ground planes
  - Local decoupling capacitors should connect to power and ground pins of all active components with minimal inductance.
- For boards with ground plane, but no power plane
  - Local decoupling capacitors should be located as close to power pin as possible.
- For boards with power and ground planes >20 mils apart
  - Local decoupling capacitors should be located as close to the active device as possible (near pin attached to most distant plane).
- For boards with power and ground planes <20 mils apart
  - All decoupling capacitors are global and should be connected to planes with minimal inductance.


SMC
FDC37C677
A9731-A54910
6C7727.1
©PHOENIX 1995 MK42
U7B1
C7B1
RP7C1
RP7C2
RP7C3
RP7C6
C7C1
C7C2
R7C1
R7C2
R7C3
R7B2
U8C1
16K
701
220
LEARN EMC

Inductance of Connections to Planes

On boards with closely spaced power and ground planes:

Generally speaking, 100 decoupling capacitors connected through 1 nH of inductance will be as effective as 500 decoupling capacitors connected through 5 nH of inductance.



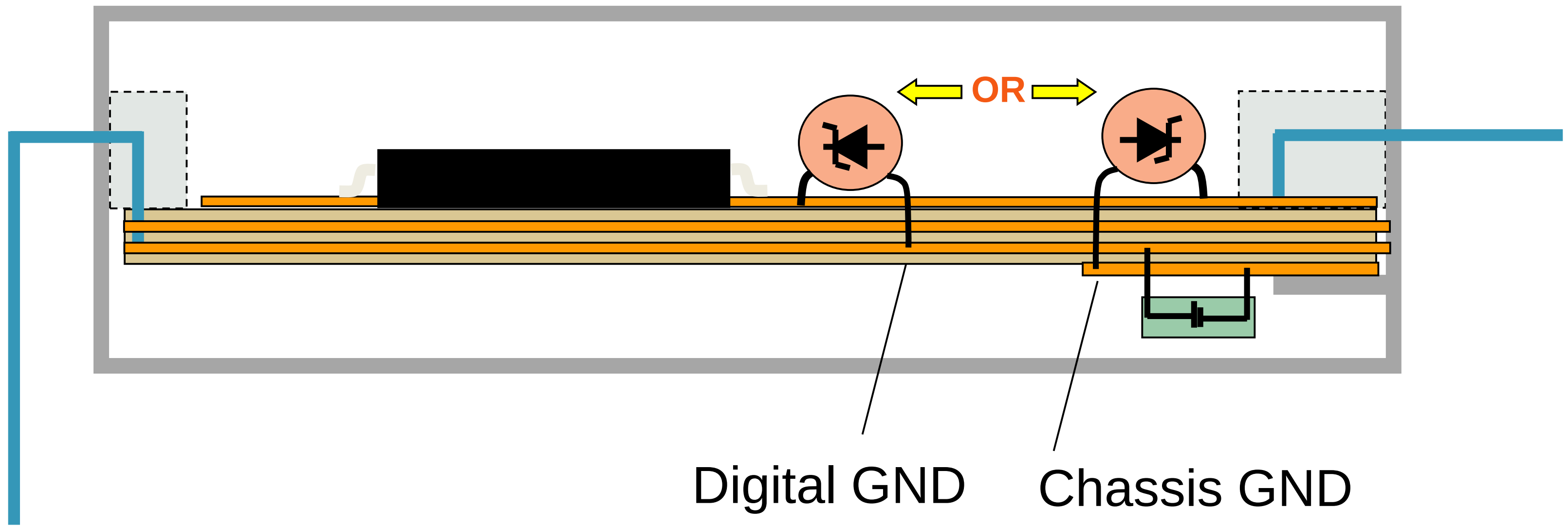
Automotive and Industrial Design for EMC

- To guarantee that your design will meet its EMC requirements the first time, you must:

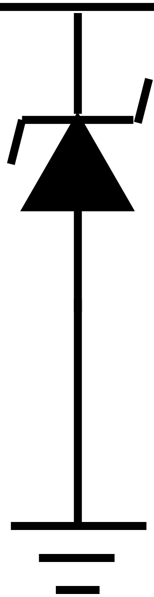
Provide adequate transient protection!

Protecting Components from Transients on the Harness

Design Exercise: Where should the transient protection be grounded?



Transient Protection Options



Diodes	0.5 volts to ~10 volts Lowest Energy High Capacitance (10’s of pF) Usually fail short Voltage limiting device
Varistors	0.5 volts to 10’s of volts Low Energy Higher Capacitance (10’s of pF) Usually fail short Voltage limiting device
Thyristors	0.5 volts to 10’s of volts Low Energy Higher Capacitance (10’s of pF) Usually fail short Voltage limiting device
Gas Discharge Tubes	10’s of volts to 1000’s of volts High Energy Low Capacitance (< 1 pF) Fail open Crowbar device

Review

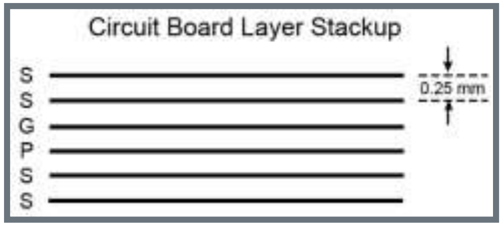
- Control your transition times
- Know how your currents return to their source
- Recognize the 4 (not 2) possible coupling mechanisms
- Identify your antennas
- Identify your sources
- Don’t rely on EMC design guidelines
- Don’t gap your ground planes
- There are no shielded enclosures in the automotive world
- Provide adequate power bus decoupling
- Provide adequate transient protection

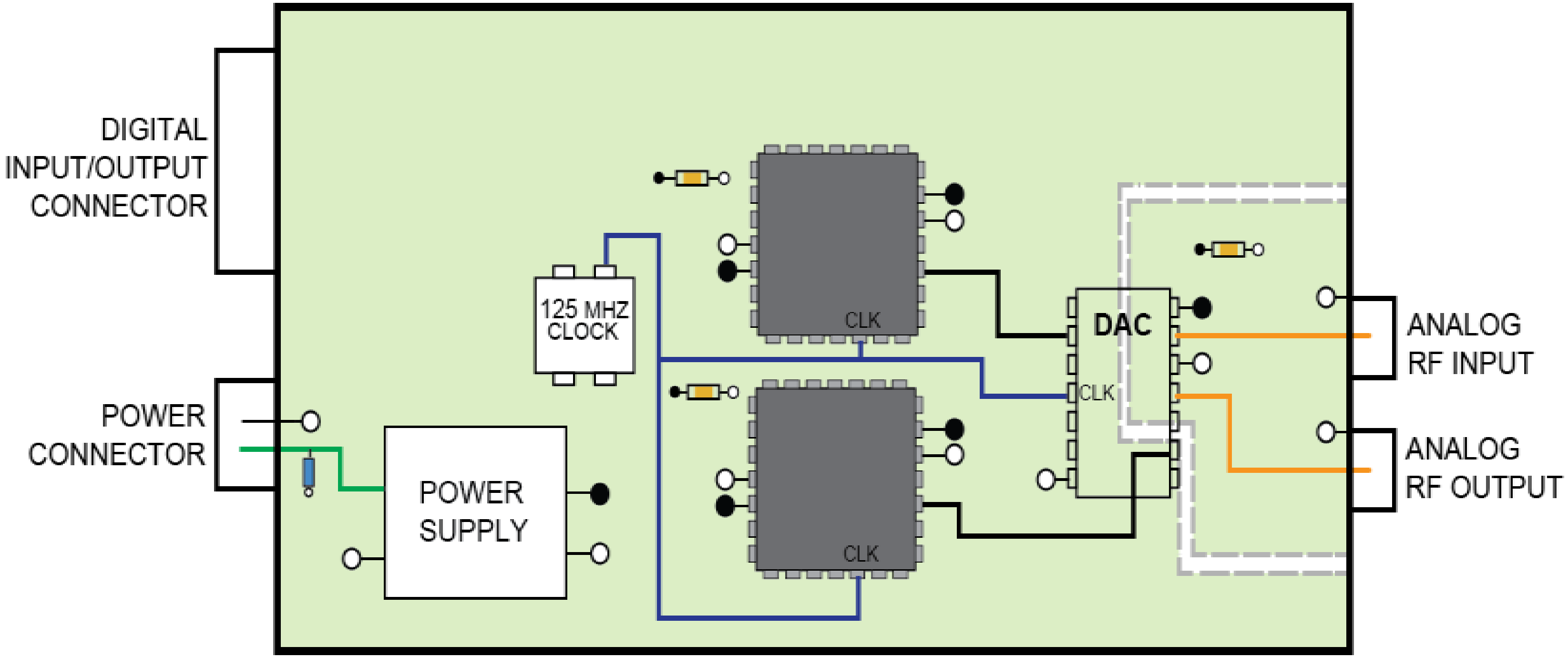
EMC Design Review

- The goal of the design review is to ensure that the system meets its EMC requirements the first time it is tested.
- Identify your antennas(ports) and coupling paths for each EMC test.
- Identify your ground structure.
- Trace the current paths.
- Control your risetimes.
- Provide adequate decoupling.
- Don’t blindly follow design rules, but establish rules appropriate for your application and evaluate all exceptions.

Be methodical!

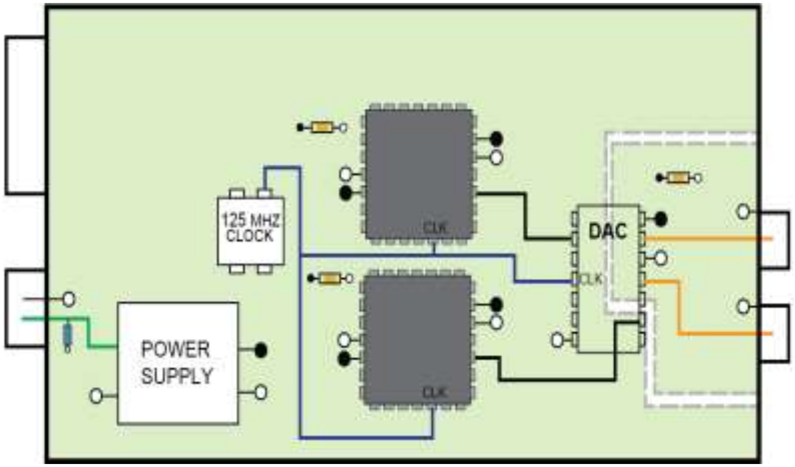
Circuit Board Layout for EMC: Example 3



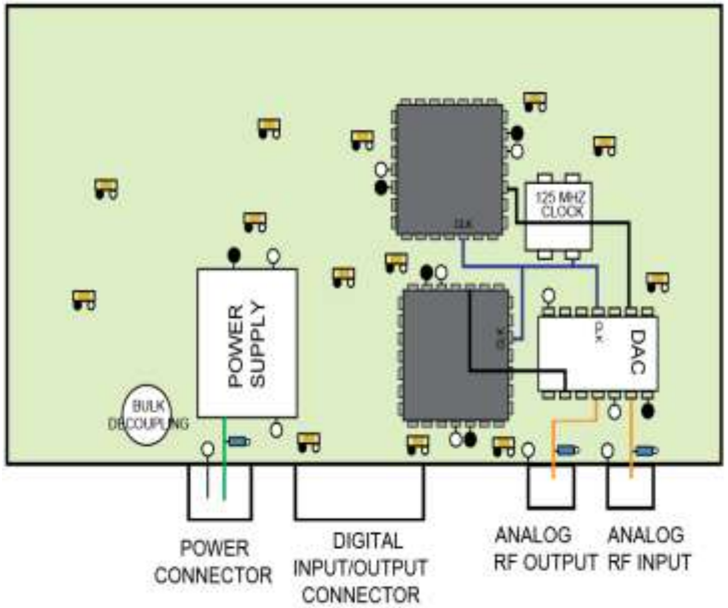


Comparison

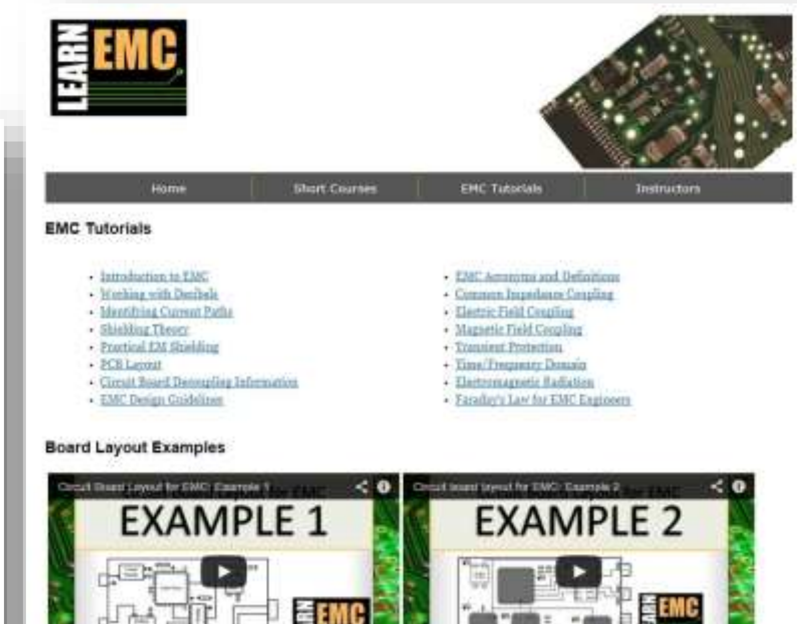
Original Layout



New Layout



For More Information



http://www.LearnEMC.com/

http://www.cvel.clemson.edu/

www.Freescale.com