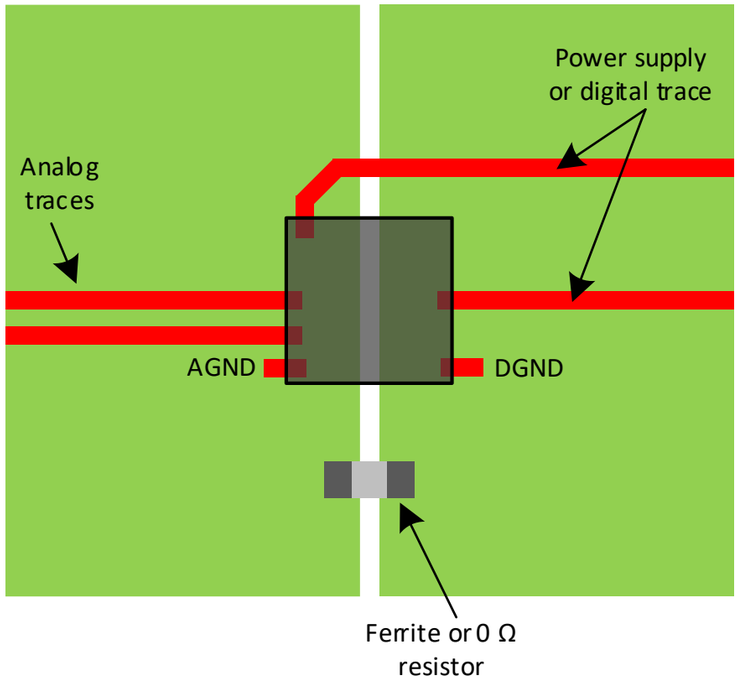
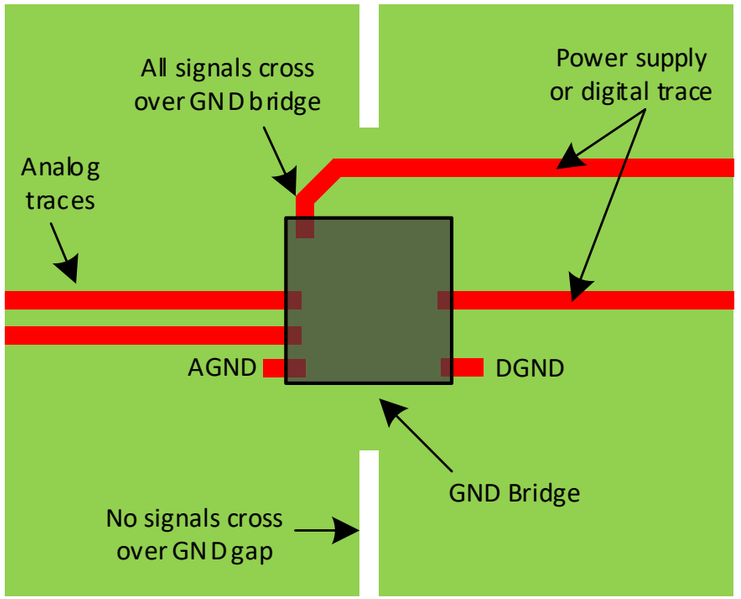
Split plane?

Worst Grounding Design Choice



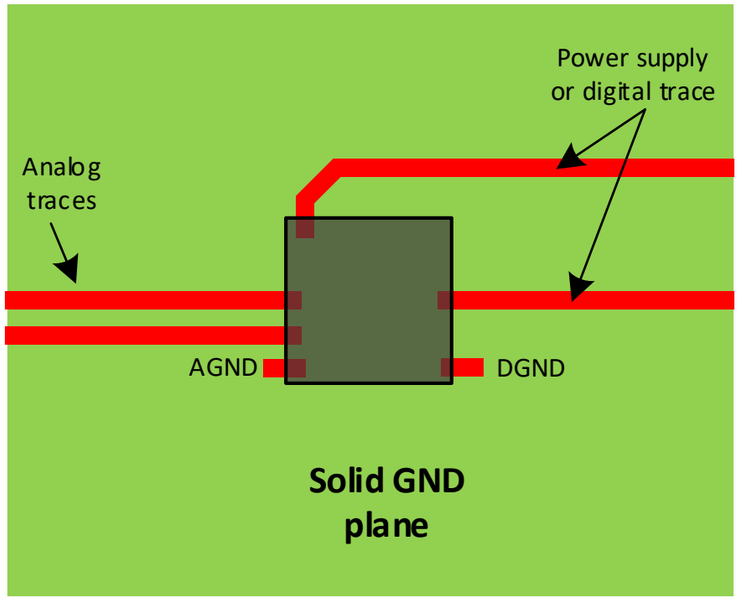
Digital communications errors likely. Significant RF emissions. Ferrite is worse than zero ohm.

Better Grounding Design Choice



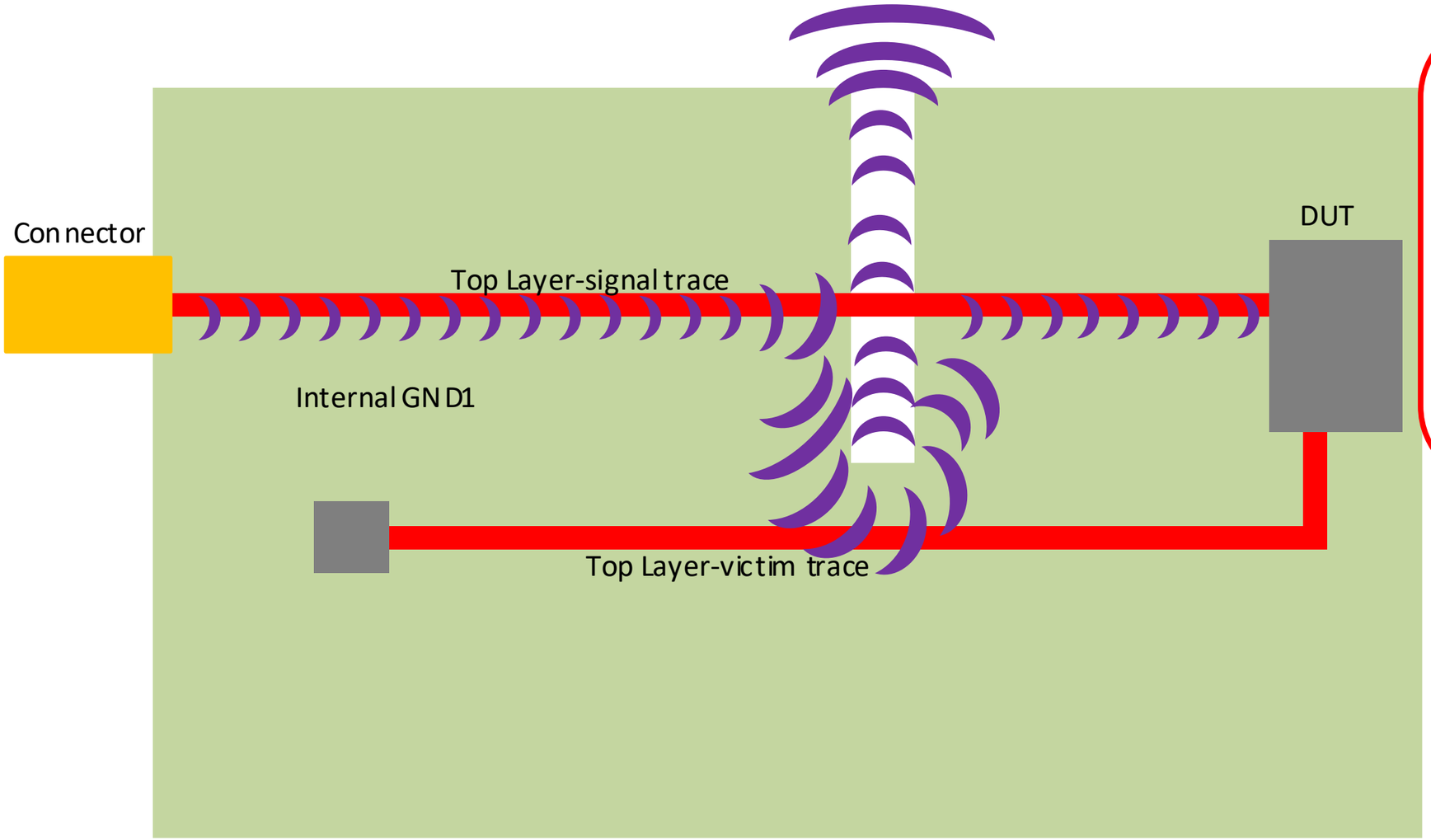
Minimal issues as long as traces cross over bridge. Slots may act as antenna. Separate planes form dipole antenna.

Best Grounding Design Choice



Best performance for both analog and digital.

Slot in GND plane causes RF emissions

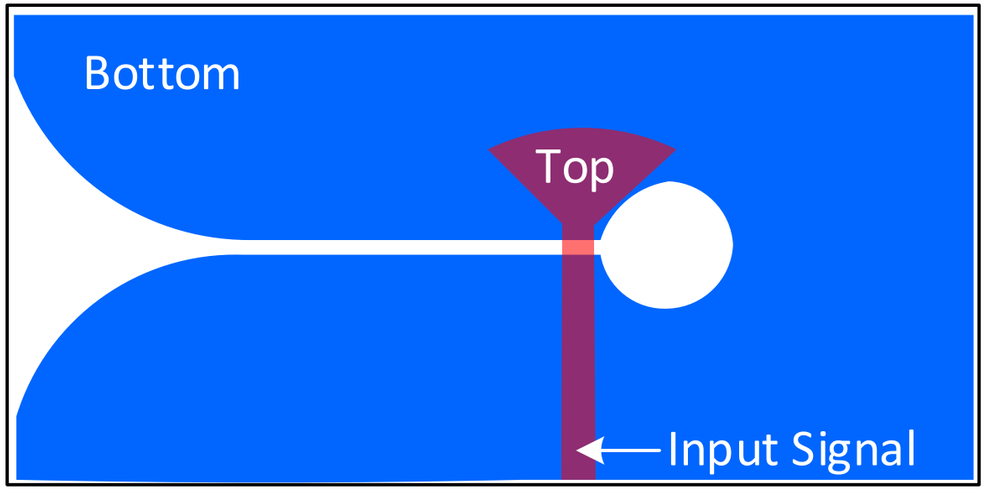


Problem with Slot in return path

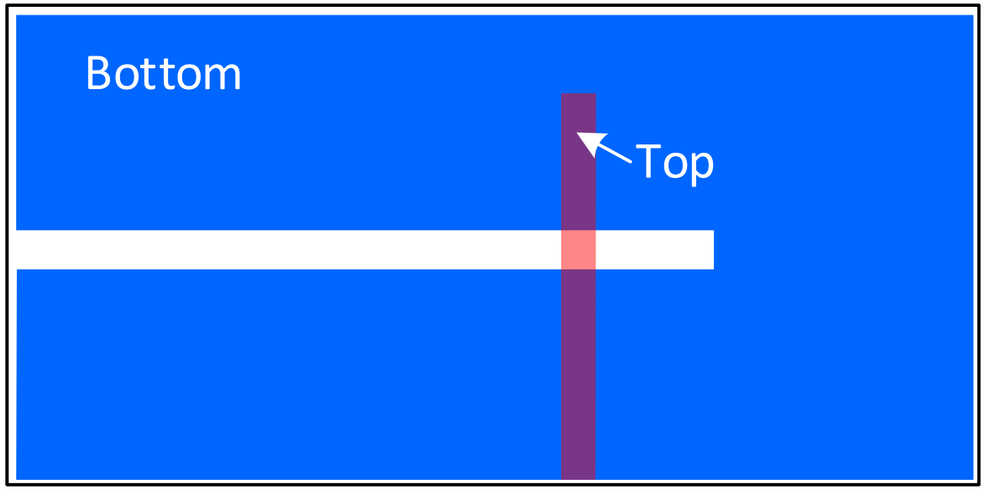
- Return current needs to find a new path when the return plane has a discontinuity
- Some current travels around the slot and causes crosstalk in adjacent “victim” trace.
- Slot acts like an antenna and transmits RF off board and onto adjacent circuits.

Reciprocity: Transmitters can also be Receivers

Vivaldi PCB Antenna



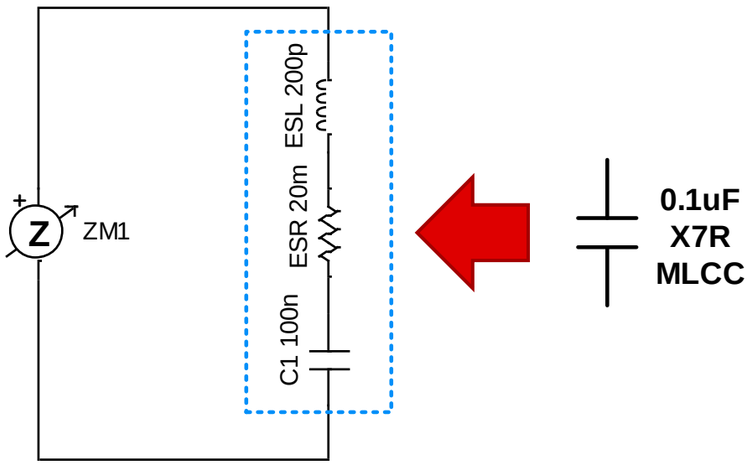
Unintended PCB Antenna
Transmits as well as receives



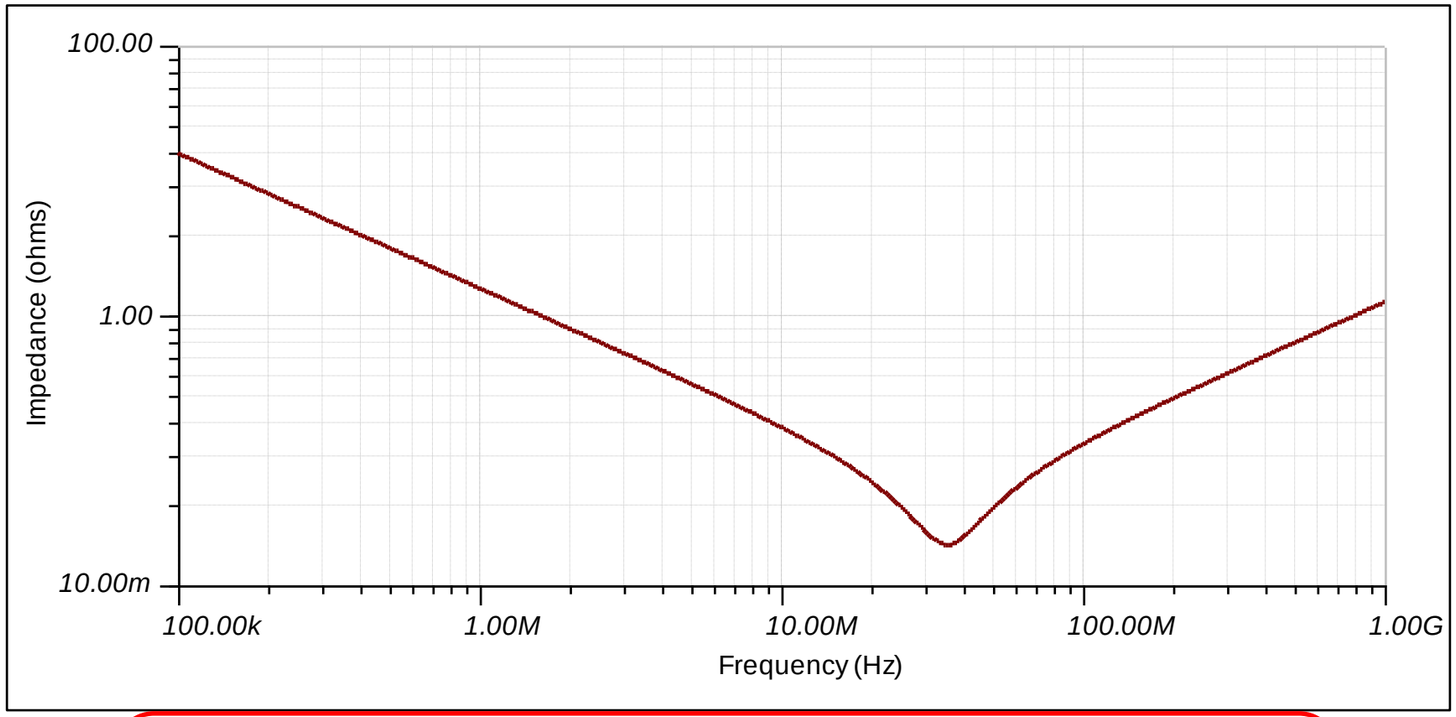
If you wanted to create an Antenna on a PCB the above two layer board design is one approach.

Running low level analog over a gap in the GND return can make it susceptible to pick up RF emissions

Model of a practical capacitor

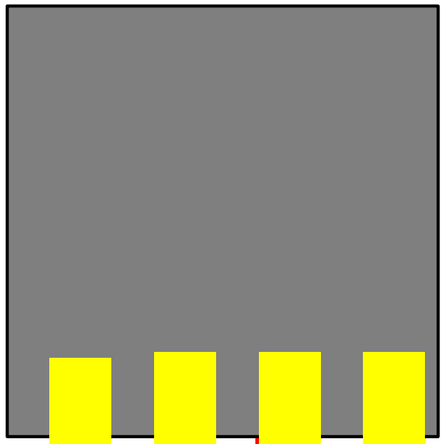


0.1uF
X7R
MLCC

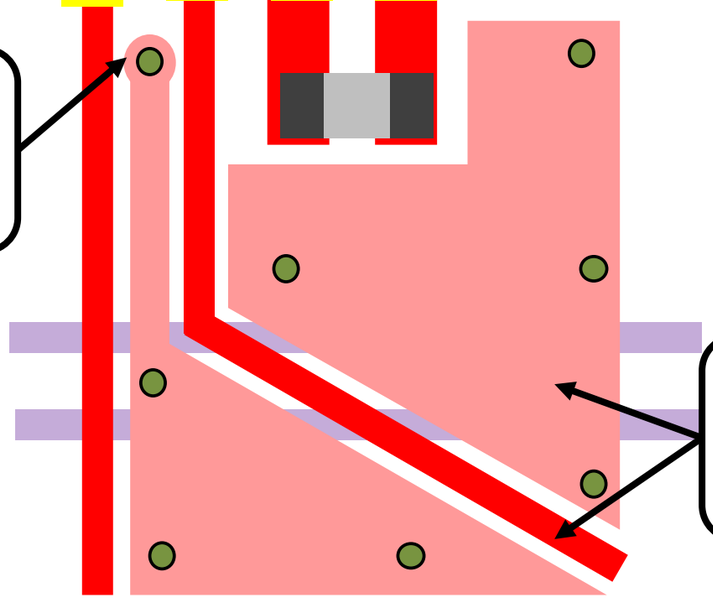


At frequencies above 30MHz this capacitor is acting like an inductor so it is not an effective filter at high frequencies.

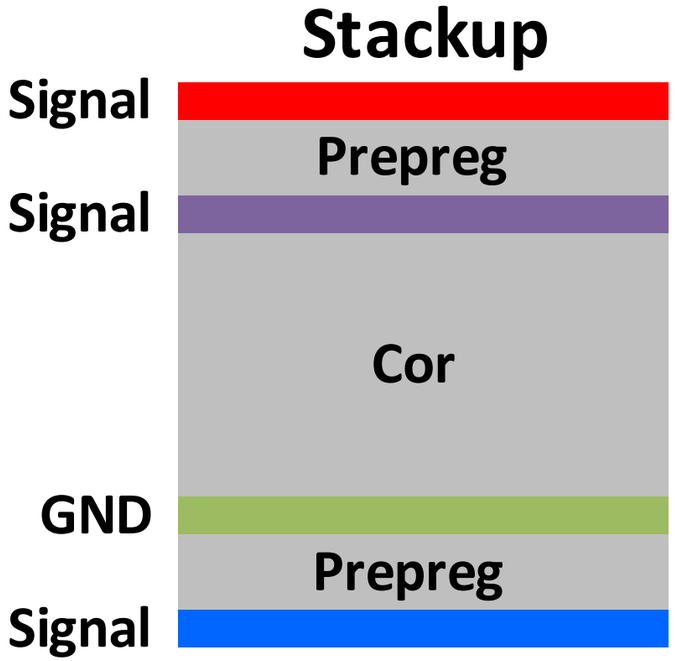
To pour or not to pour … That is the question



Terminate all gnd traces with via. Don't leave it float



GND pour acts as GND return for adjacent traces. No plane in this example stackup.



- **Generally advise against!**
- If signal layer is adjacent to GND plane pour is not needed.
- Pour may be helpful for 1 or 2 layer boards to provide good GND return when a plane is not available
- Never pour a copper without vias to GND
- Place vias every 1/10 wavelength of maximum expected frequency.
- Lack of GND via can translate pour area into antenna

What is wrong here?