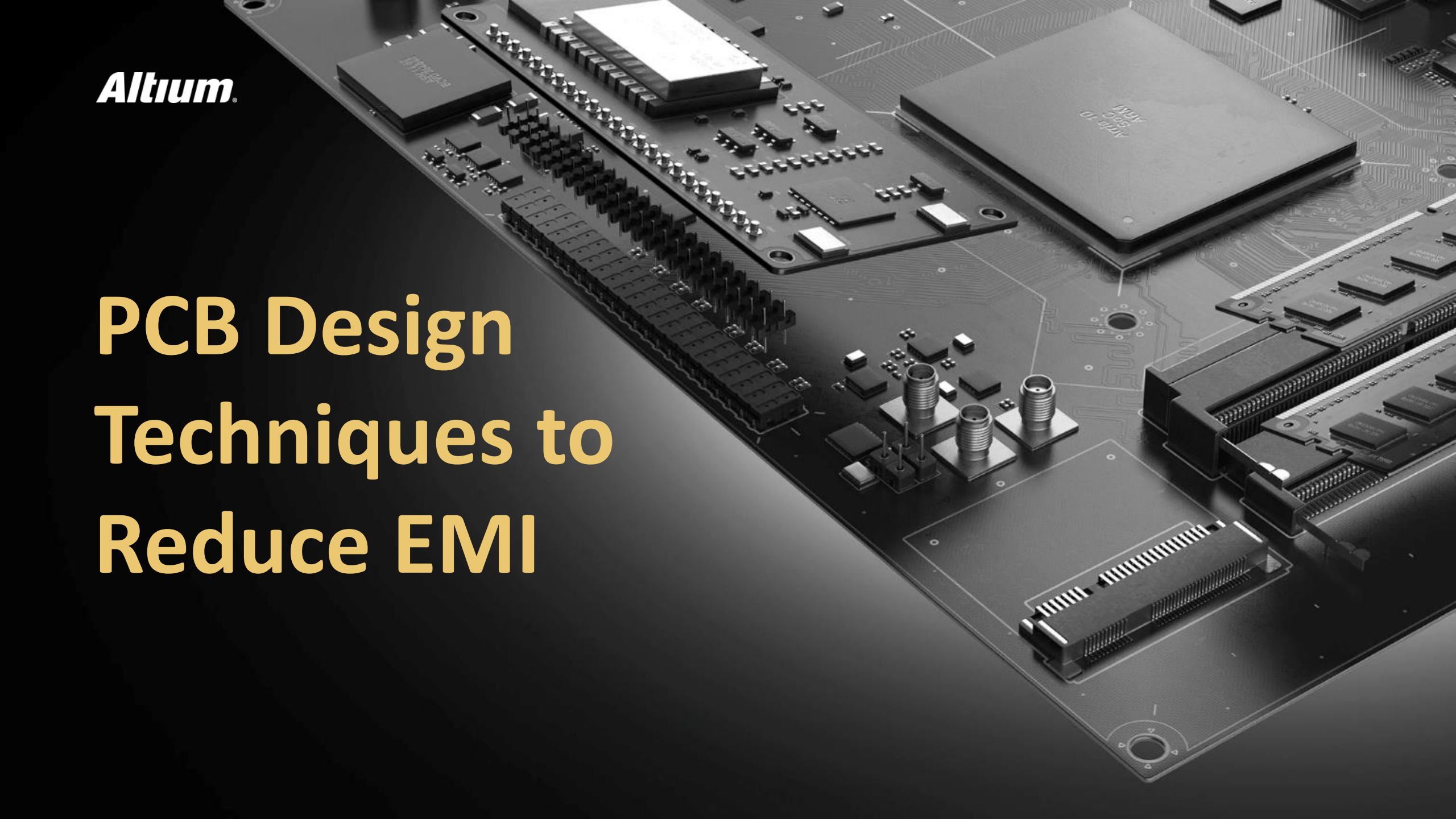
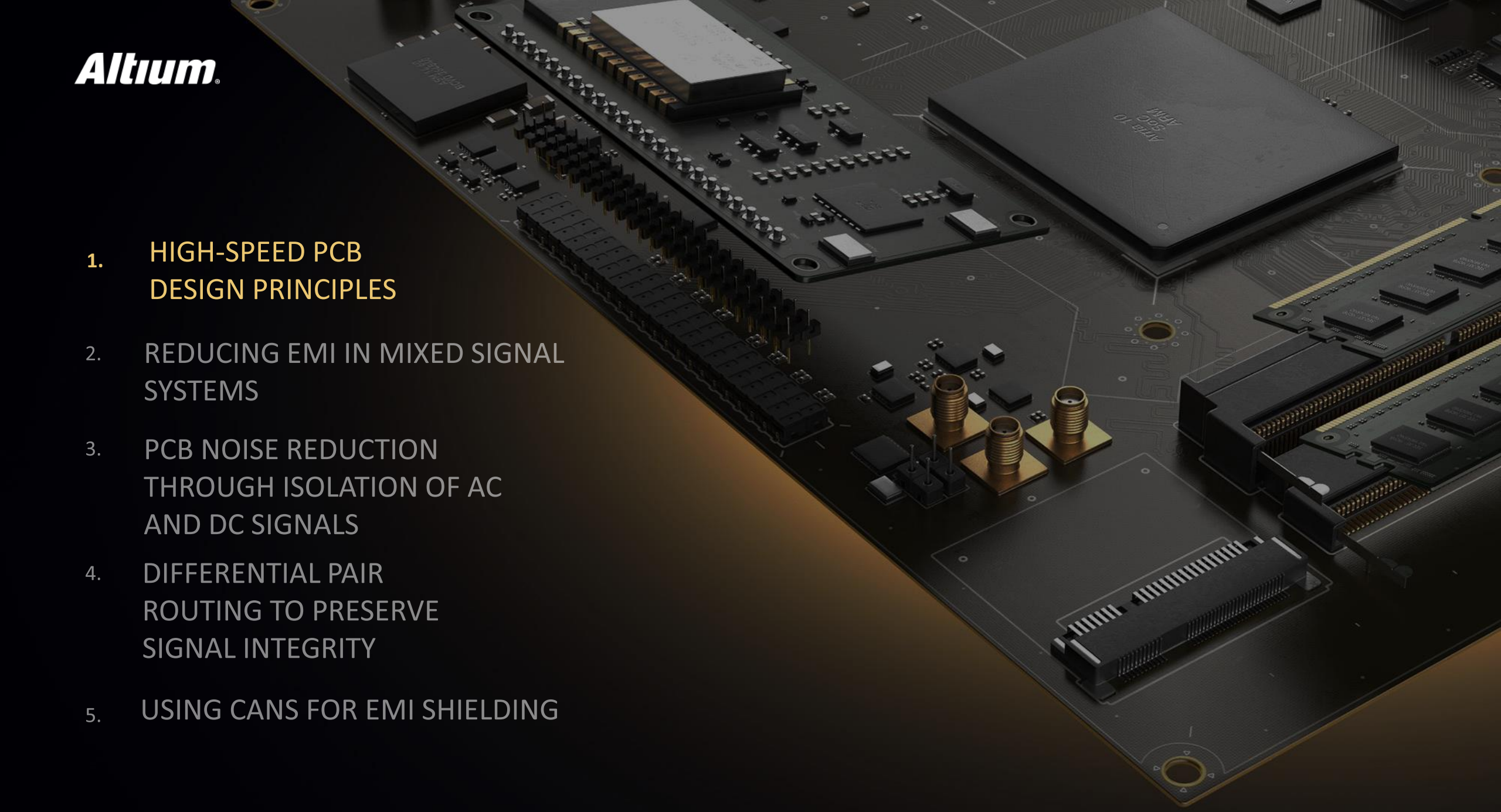
Altium

PCB Design Techniques to Reduce EMI



Altium



1. HIGH-SPEED PCB DESIGN PRINCIPLES
2. REDUCING EMI IN MIXED SIGNAL SYSTEMS
3. PCB NOISE REDUCTION THROUGH ISOLATION OF AC AND DC SIGNALS
4. DIFFERENTIAL PAIR ROUTING TO PRESERVE SIGNAL INTEGRITY
5. USING CANS FOR EMI SHIELDING

Altium

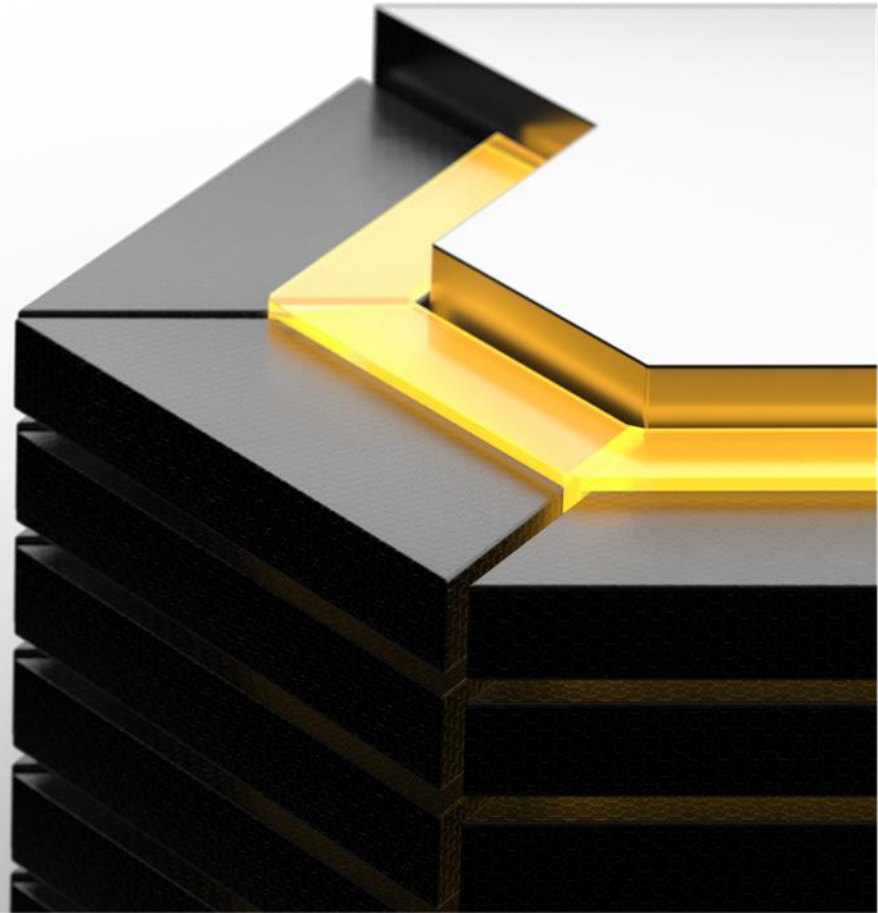
1. HIGH-SPEED PCB DESIGN PRINCIPLES

Altium

Just the Basics

To avoid problems with EMI on your high-speed PCB, keep traces short and direct, but not at the cost of crossing gaps in the ground plane.

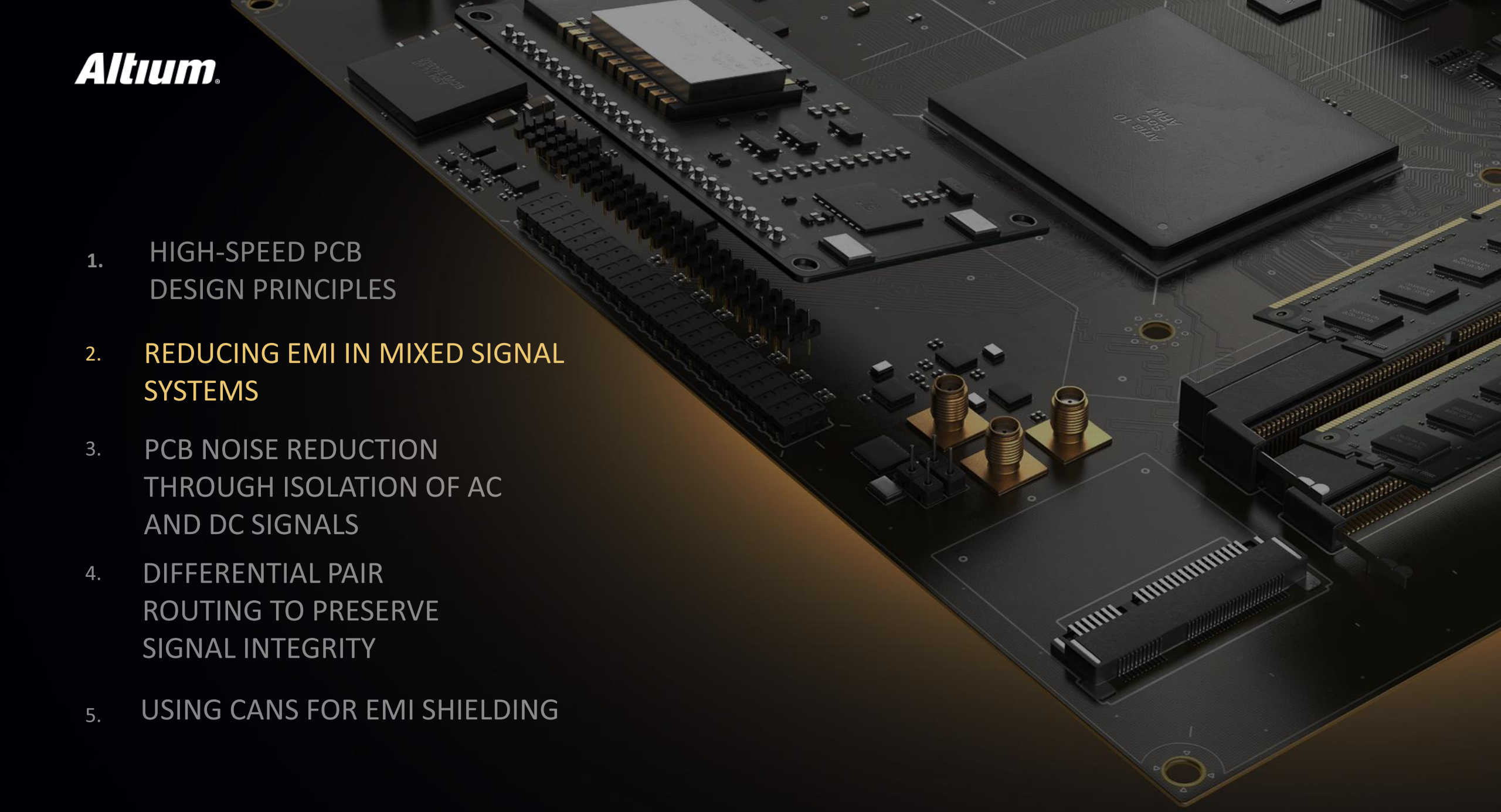
- Longer traces risk greater EMI, cost more money, and eat up more real estate
- Color coding can help keep track of components and traces
- Trace direct and think before you route



[See original content here]

Altium

1. HIGH-SPEED PCB DESIGN PRINCIPLES
2. REDUCING EMI IN MIXED SIGNAL SYSTEMS
3. PCB NOISE REDUCTION THROUGH ISOLATION OF AC AND DC SIGNALS
4. DIFFERENTIAL PAIR ROUTING TO PRESERVE SIGNAL INTEGRITY
5. USING CANS FOR EMI SHIELDING



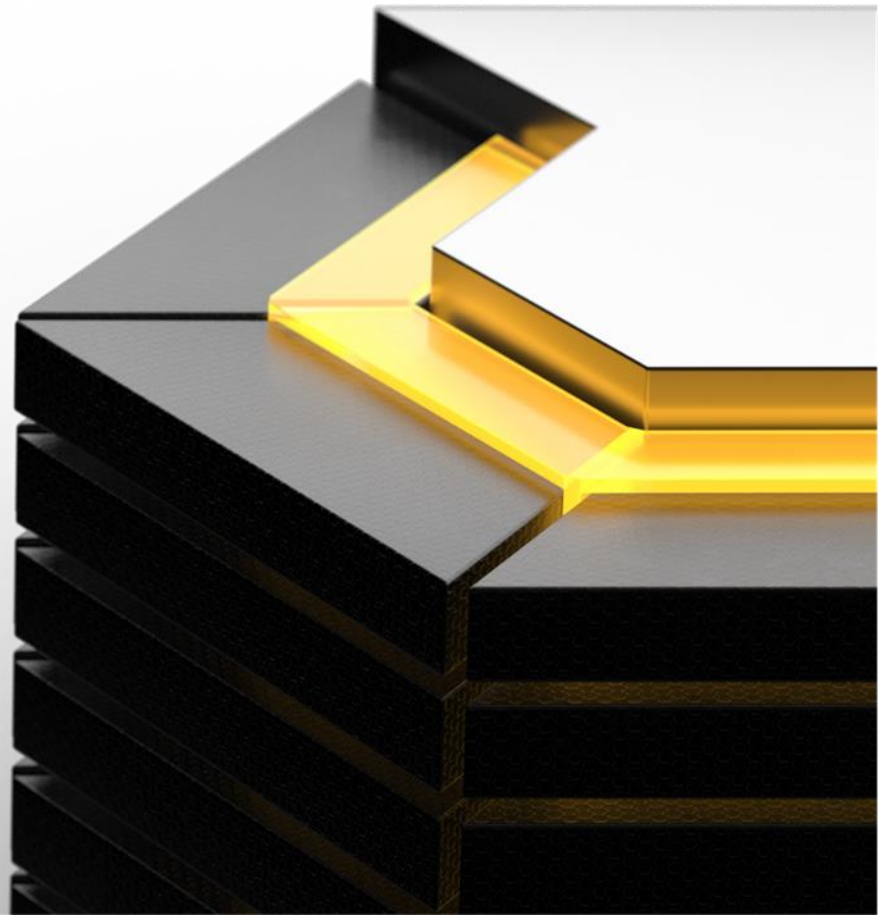
Altium

2. REDUCING EMI IN MIXED SIGNAL SYSTEMS

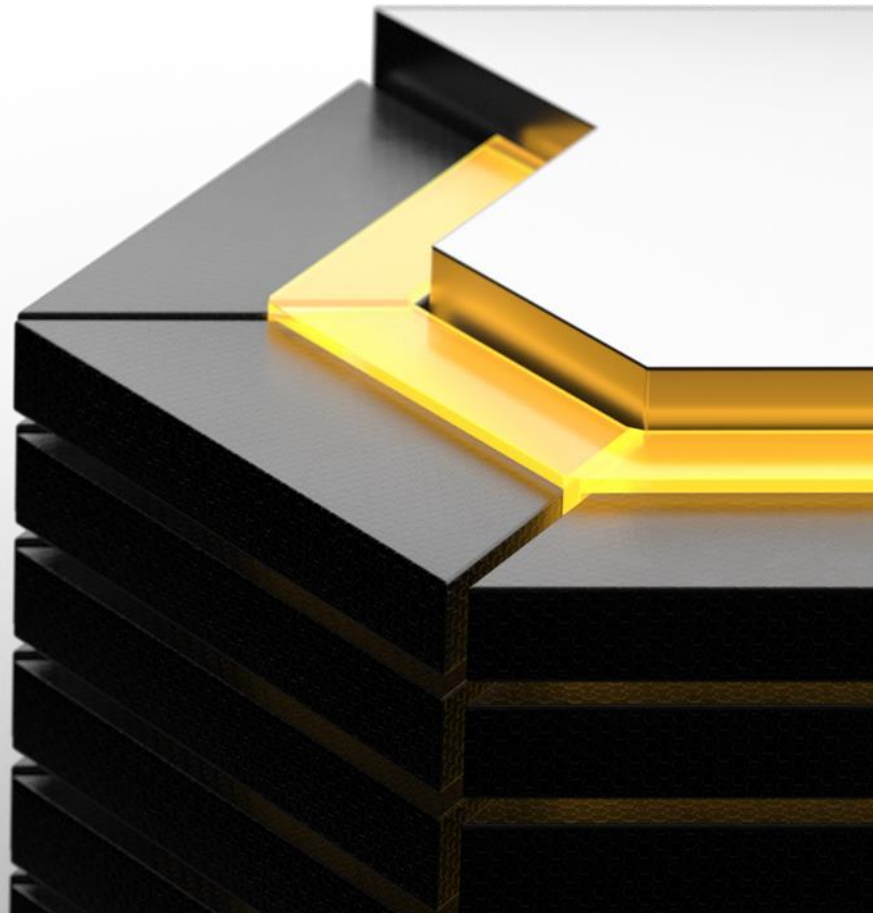
Using Proper PCB Ground Designs

Designing a mixed signal grounding PCB is challenging, but a few best practices can help simplify the process.

- Bus lines are not a great option
- Grounding grids are OK for smaller PCBs (but be careful about crosstalk)
- Grounding planes are the best option for mixed grounding
- Merging grounding planes or using a single one is acceptable if the AC/DC paths do not cross
- In the case of two grounding planes, connect using a ferrite bead or Schottky diodes
- Be careful about introducing errors and different references for certain parts of the circuit

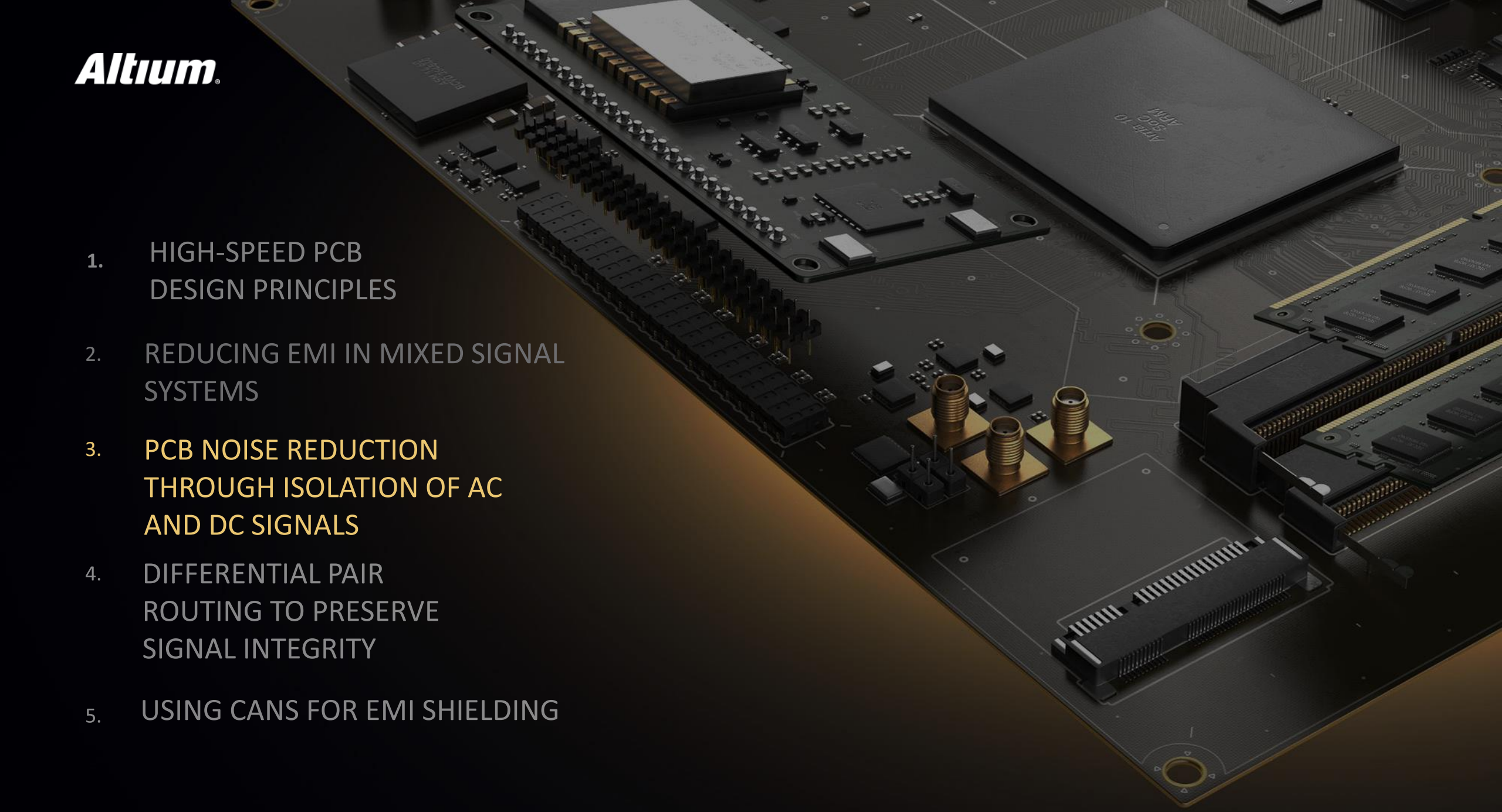


- Star grounds
- Use a ground plane with gaps to separate return currents
- Use a ground plane without gaps and very carefully checked return current paths



See original content here

Altium



1. HIGH-SPEED PCB DESIGN PRINCIPLES
2. REDUCING EMI IN MIXED SIGNAL SYSTEMS
3. PCB NOISE REDUCTION THROUGH ISOLATION OF AC AND DC SIGNALS
4. DIFFERENTIAL PAIR ROUTING TO PRESERVE SIGNAL INTEGRITY
5. USING CANS FOR EMI SHIELDING

Altium

3. PCB NOISE REDUCTION THROUGH ISOLATION OF AC AND DC SIGNALS

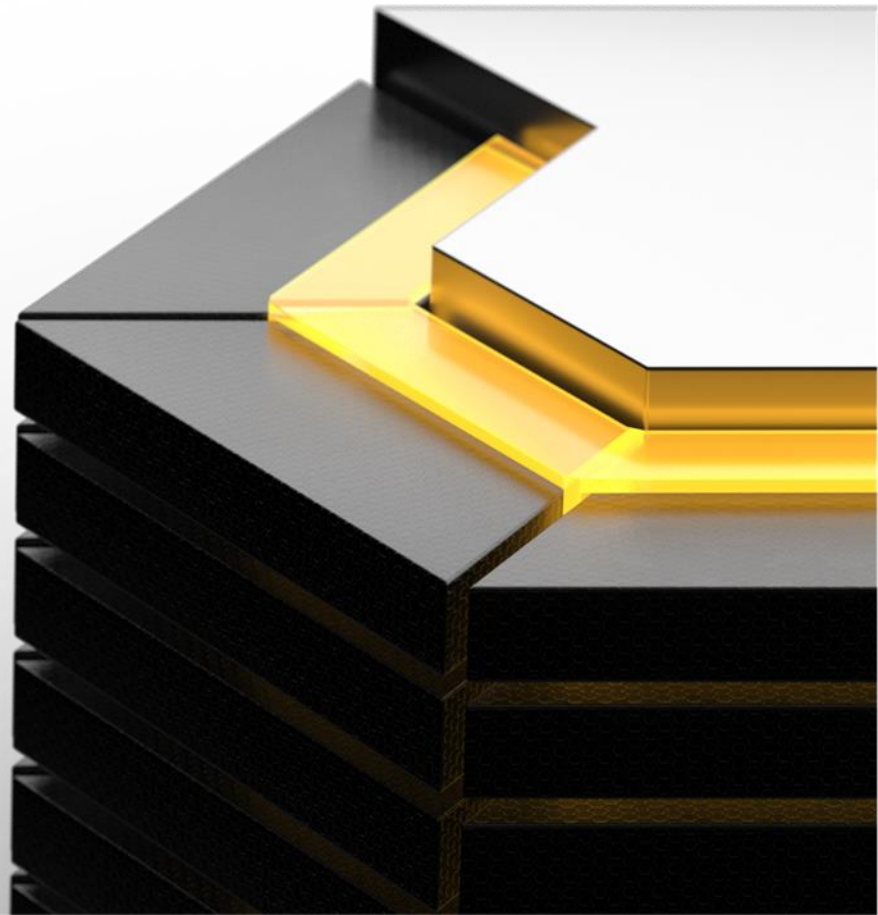
Altium

Guide to Isolating Circuits

On a board that uses both AC and DC power, isolating these circuits is critical. To manage electromagnetic interference (EMI) resulting from having AC and DC power on the same board, consider these 5 things:

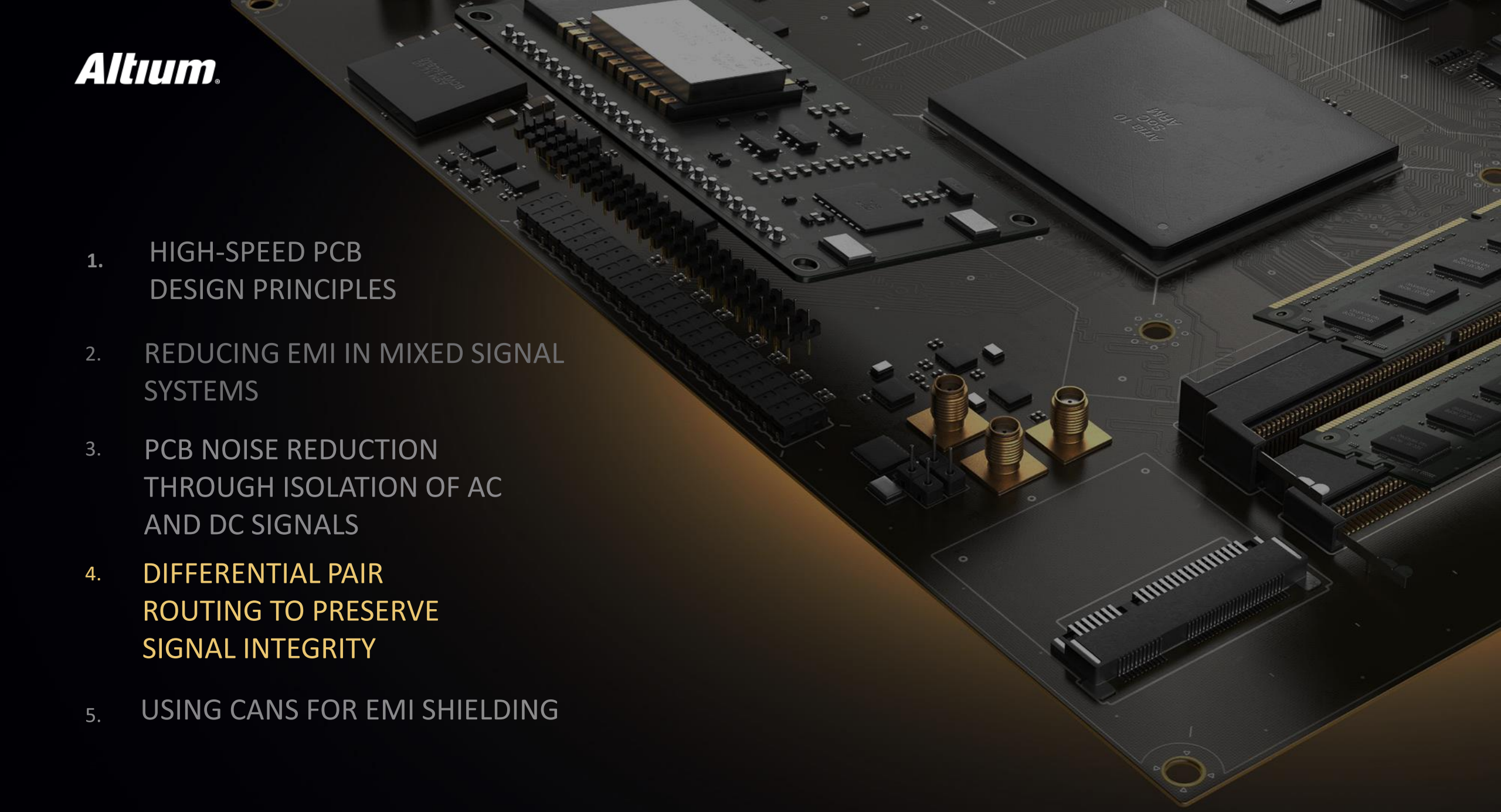
- *Shielding* may increase cost or weight of board
- *Separation* is easy with simple circuits, but more difficult with complex ones
- *Grounding* shouldn't neglect the return paths and ensure wires aren't crossed
- *Separating* power supplies is critical, if not necessarily convenient
- *Don't bridge gaps*, this isn't a good

[See original content here]



Altium

1. HIGH-SPEED PCB DESIGN PRINCIPLES
2. REDUCING EMI IN MIXED SIGNAL SYSTEMS
3. PCB NOISE REDUCTION THROUGH ISOLATION OF AC AND DC SIGNALS
4. DIFFERENTIAL PAIR ROUTING TO PRESERVE SIGNAL INTEGRITY
5. USING CANS FOR EMI SHIELDING



Altium

4. DIFFERENTIAL PAIR ROUTING TO PRESERVE SIGNAL INTEGRITY

Altium

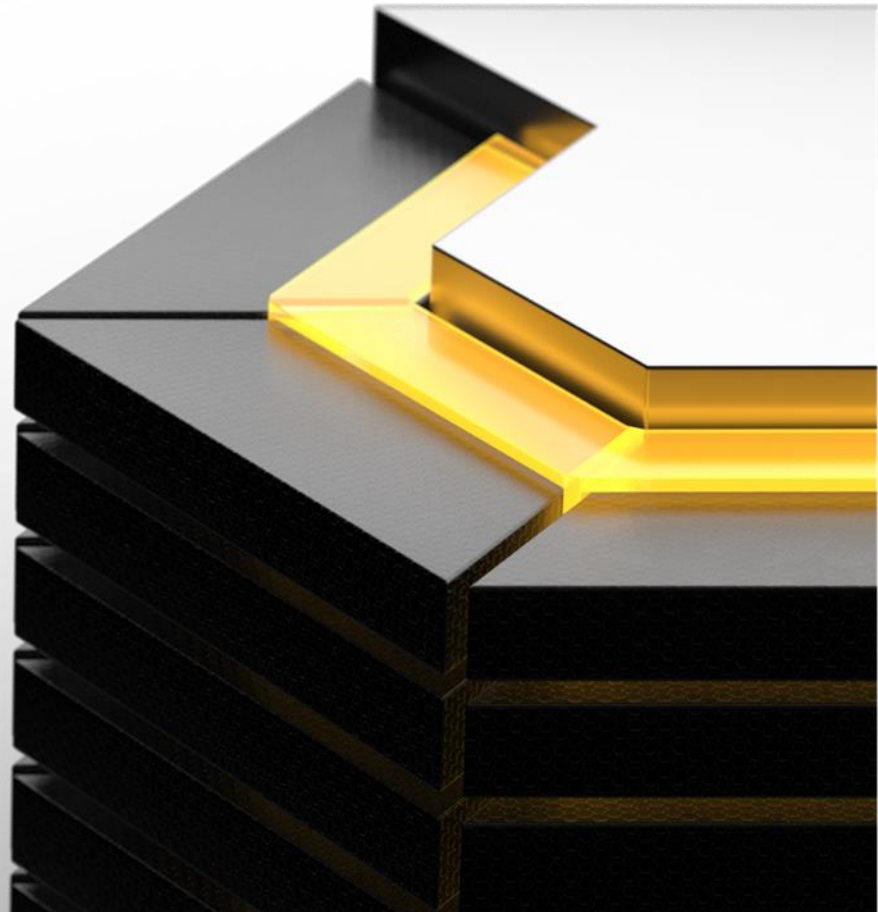
How To Preserve Signal Integrity

Correctly routed differential pairs are critical to signal integrity and general functionality. There are a few ways to make sure differential pairs are routed with timing in mind:

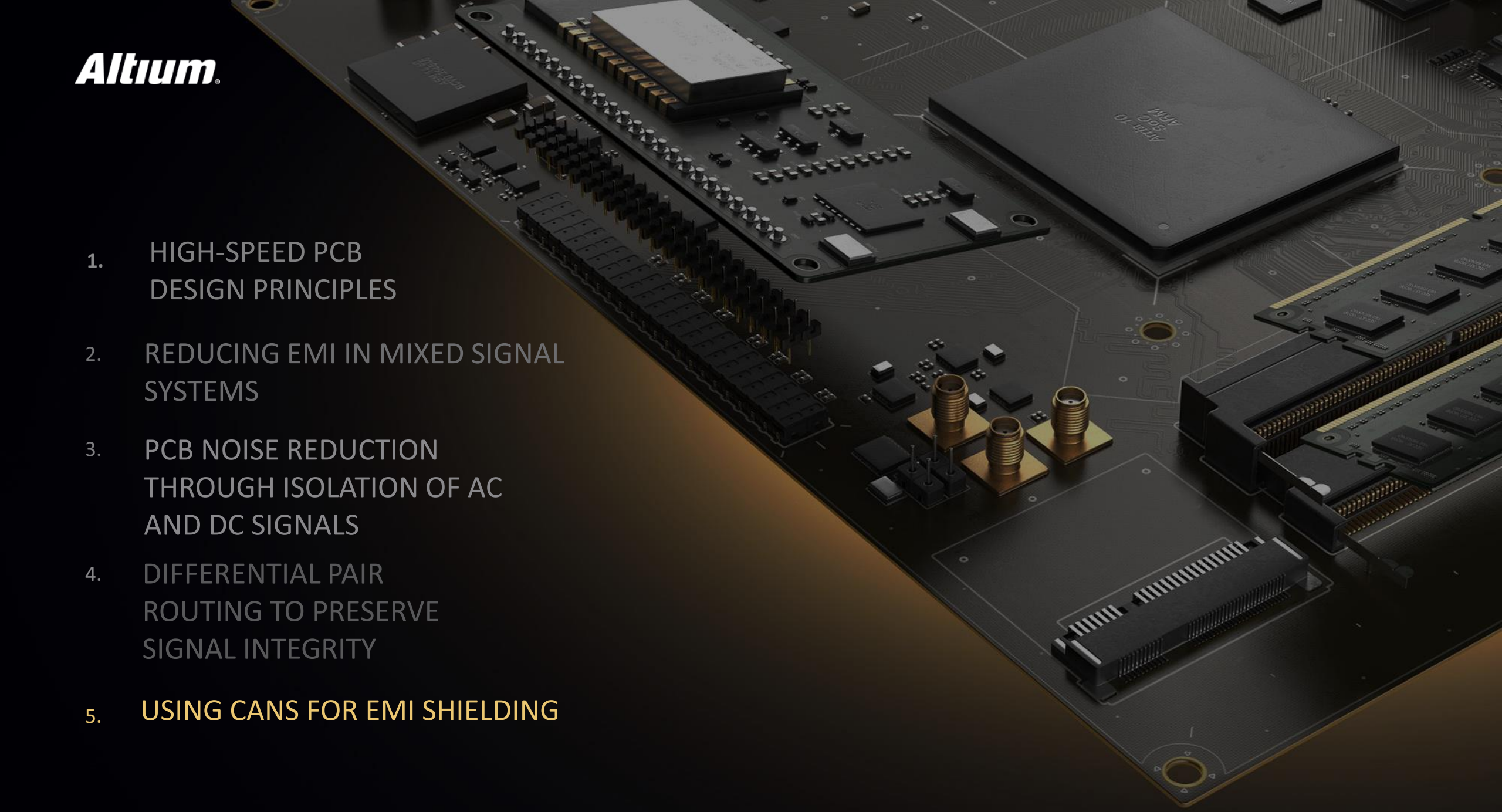
- *Trace length matching* is top priority
- *Parallel routing* cancels out radiated EMI and assists with trace length matching
- *Electrical clearing and creepage* can reduce EMI by separating differential pairs
- *No sharp turns*, instead route straight if possible, smooth curves if necessary
- *Via geometry* guarantees at least a small amount of signal degradation

PCB design software can automatically check for some of these rules to optimize your time and brainpower.

[See original content here]



Altium



1. HIGH-SPEED PCB DESIGN PRINCIPLES
2. REDUCING EMI IN MIXED SIGNAL SYSTEMS
3. PCB NOISE REDUCTION THROUGH ISOLATION OF AC AND DC SIGNALS
4. DIFFERENTIAL PAIR ROUTING TO PRESERVE SIGNAL INTEGRITY
5. USING CANS FOR EMI SHIELDING

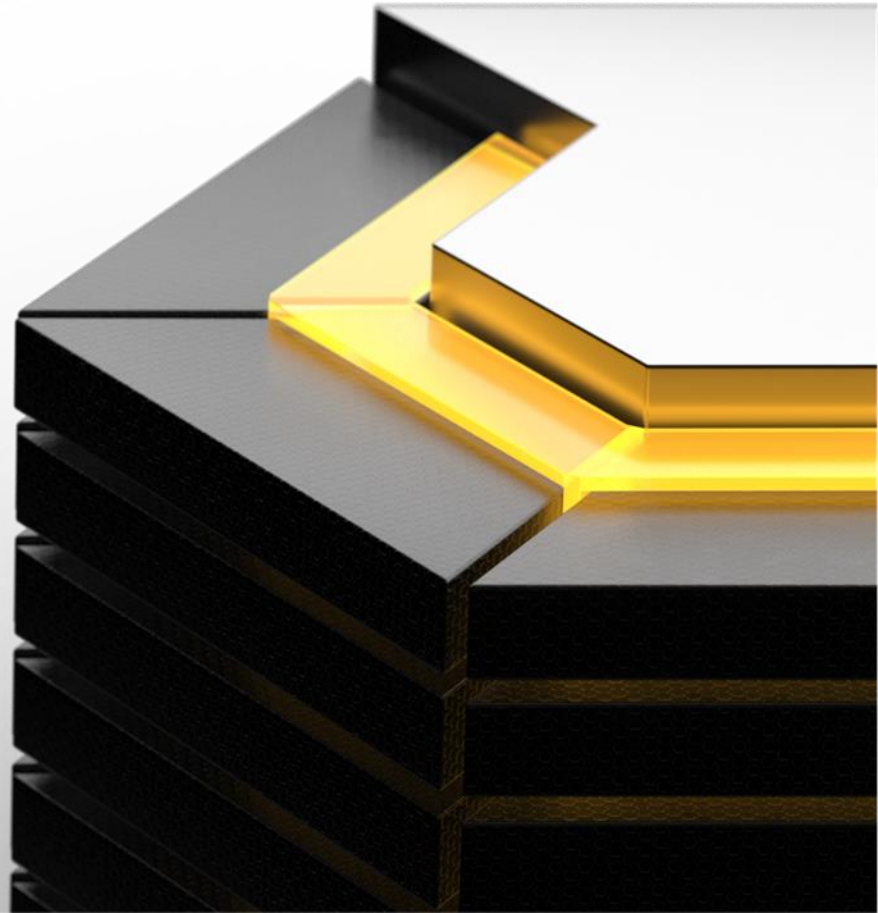
Altium

5. USING CANS FOR EMI SHIELDING

The Next Step To Minimizing Emissions

Sometimes even the best design can leave sensitive components vulnerable. To protect components or subassemblies, consider using EMI shields (also called cans, cages, covers or lids). These metal boxes attach to your PCB to enclose the circuitry and reduce EMI. While they should never take the place of good design practices like short traces, proper ground, and component placement, they are a great next step in minimizing emissions.

[See original content here]



Altium

Thanks for your attention!