

PCB EMC Guidelines

- Minimize signal current loop areas
- Don't locate circuitry between connectors
- Control transition times of digital signals
- Provide a solid (not gapped) signal return plane

D. Beetner Slide 1

Component Placement

- Locate connectors on one edge or one corner of the board
- A device that communicates off board should be located as closely as possible to the connector it uses
- Components not connected to an I/O net should be located at least 2 cm from I/O nets and connectors
- All off-board communications from a single device should be routed through the same connector
- Clock drivers should be located next to clock oscillators

D. Beetner Slide 2

Trace Routing

- All power planes and traces should be routed on the same layer
 - Locate power planes at least 3 mm apart
- Critical signal traces should be buried between power/ground planes
- No trace unrelated to I/O should be located between a device using I/O and the connector
- Signals with high-frequency content should not be routed beneath components used for board I/O

D. Beetner Slide 3

Trace Routing

- Critical nets between planes should be routed at least 2X from the board edge, where X is distance from trace to return plane
- On a board with power and return planes, connections to the planes should be made directly with vias – no traces
- On boards with multiple return planes, all vias connected to one signal return plane should be connected to the other return planes

D. Beetner Slide 4

Power Bus Decoupling

- No power planes
 - Provide a bulk decoupling capacitor and at least one local decap for each component
 - Size bulk decap for transient current needs of entire board. Typically 1-10 times size of all local decaps.
 - Bulk decap should be located close to the power source (connector/battery/etc)

D. Beetner Slide 5

Power Bus Decoupling

- No power planes
 - Local decaps should be placed between power and return pins. The device/component loop area should be minimized
 - A general rule, 2 decoupling capacitors with the same value are better than 1 capacitor with twice the value.

D. Beetner Slide 6

Power Bus Decoupling

- Closely spaced power planes (<0.3 mm)
 - Use one or more large “bulk” decoupling capacitors (e.g. electrolytic) to supply large current needs at low frequencies.
 - Size bulk decap for transient current needs of entire board. Typically 1-10 times size of all local decaps.
 - Use smaller-sized (i.e. volume) “local” decaps to supply current at high frequency
 - Generally, the smaller the physical size the smaller the associated inductance

D. Beetner Slide 7

Power Bus Decoupling

- Closely spaced power planes (<0.3 mm)
 - Choose the largest value of local decoupling capacitance in a given package size.
 - Do not use capacitors smaller than the interplane capacitance
 - 10-mil spacing approx. 16pF/cm²

D. Beetner Slide 8

Power Bus Decoupling

- Closely spaced power planes (<0.3 mm)
 - Local decaps may be located anywhere reasonably close to components
 - Minimize connection inductance formed by traces and vias
 - In general, connection inductance is reduced by the square root of number of capacitors
 - Connection inductance is generally more important than capacitor value, especially at high frequency

D. Beetner Slide 9

Minimizing connection inductance

- Avoid traces to vias. Place via as close to mounting pad as possible
 - Move capacitor if necessary
 - Put via in pad
- Locate 2 vias as close together as possible
- Use multiple vias
- Mount capacitor on side of board closest to power planes
- Use wide traces

D. Beetner Slide 10

Power Bus Decoupling

- Widely spaced planes (>0.5 mm)
 - Use one or more large “bulk” decoupling capacitors (e.g. electrolytic) to supply large current needs at low frequencies.
 - Size bulk decap for transient current needs of entire board. Typically 1-10 times size of all local decaps.
 - Use smaller-sized (i.e. volume) “local” decaps to supply current at high frequency
 - Generally, the smaller the physical size the smaller the associated inductance

D. Beetner Slide 11

Power Bus Decoupling

- Widely spaced power planes (>0.5 mm)
 - Choose the largest value of local decoupling capacitance in a given package size.
 - Minimize connection inductance formed by traces and vias
 - In general, connection inductance is reduced by the square root of number of capacitors
 - Connection inductance is generally more important than capacitor value, especially at high frequency

D. Beetner Slide 12

Power Bus Decoupling

- Widely spaced power planes (>0.5 mm)
 - Location is critical
 - Locate decaps close to power/gnd pins of device trying to decouple. In particular, locate next to pin connecting to the power plane farthest from the device. Via to plane farthest from decap should be placed as close to the device via as possible (possibly sharing it), without using connection traces.

D. Beetner Slide 13