PCIe board layout recommendations

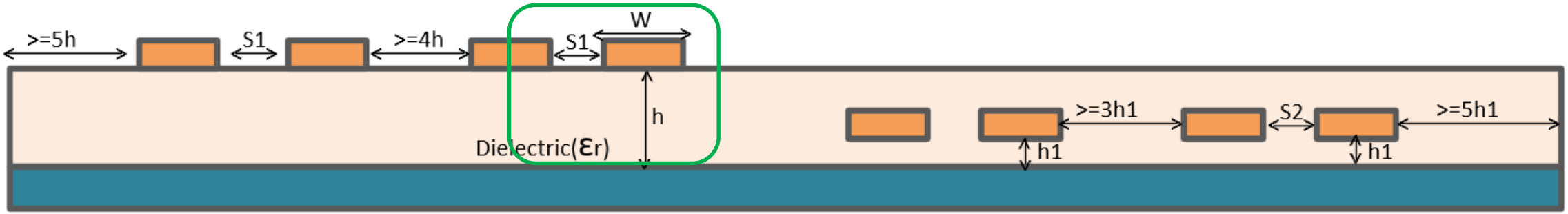
TI Precision Labs – PCIe

Prepared by Nasser Mohammadi

Presented by Nicholaus Malone

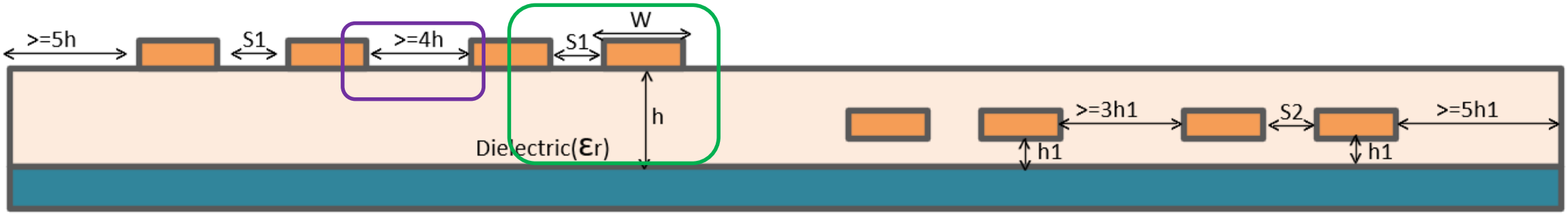
Differential board trace width and spacing

- Design trace width, height, and spacing to meet 85 ±20% Ohm for PCIe connector
  - 100 ±20% Ohm for chip to chip



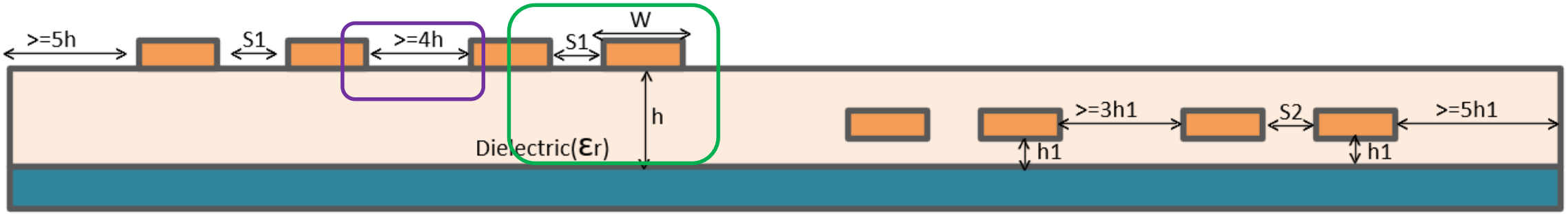
Differential board trace width and spacing

- Design trace width, height, and spacing to meet 85 ±20% Ohm for PCIe connector
  - 100 ±20% Ohm for chip to chip
- Distance between adjacent traces should be at least 4 times dielectric height(h)
  - Stripline traces should be 3 times



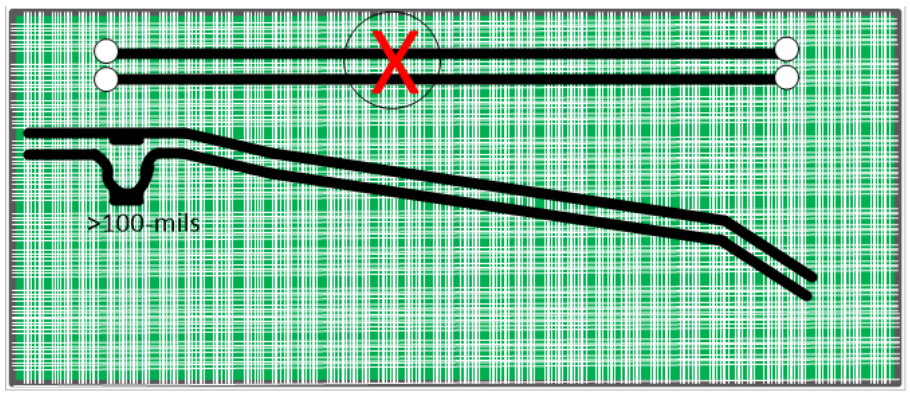
Differential board trace width and spacing

- Design trace width, height, and spacing to meet 85 ±20% Ohm for PCIe connector
  - 100 ±20% Ohm for chip to chip
- Distance between adjacent traces should be at least 4 times dielectric height(h)
  - Stripline traces should be 3 times
- Distance between PCIe pair and non-PCIe signals should be at least 4 times height
  - If non-PCIe signal has higher slew rate or level, then this distance should be 6 times height



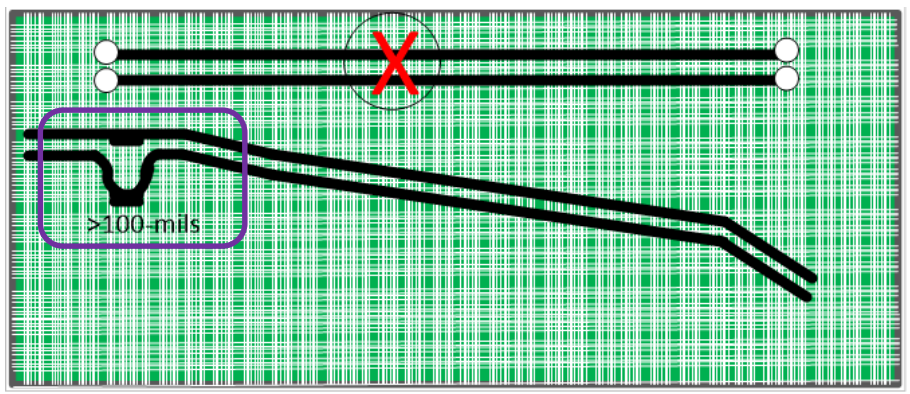
Trace length matching and trace length

- Avoid running long traces in parallel with grain of the fiber.



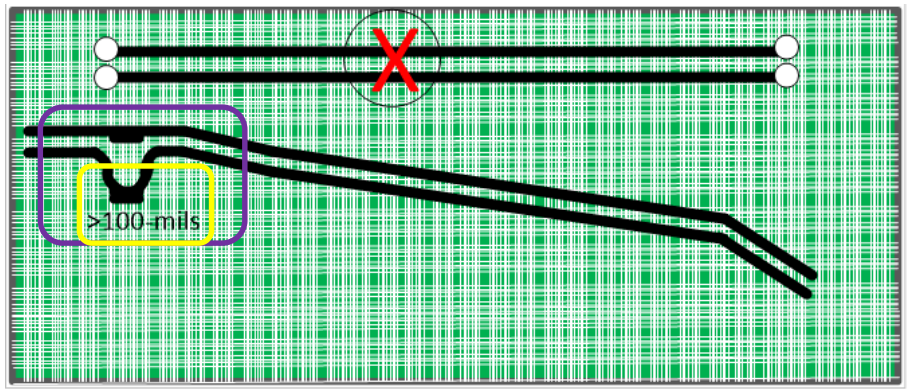
Trace length matching and trace length

- Avoid running long traces in parallel with grain of the fiber.
- Intra-pair trace should be matched to within 5-mils.
  - Trace mis-match compensation should be done at the point of mis-match.



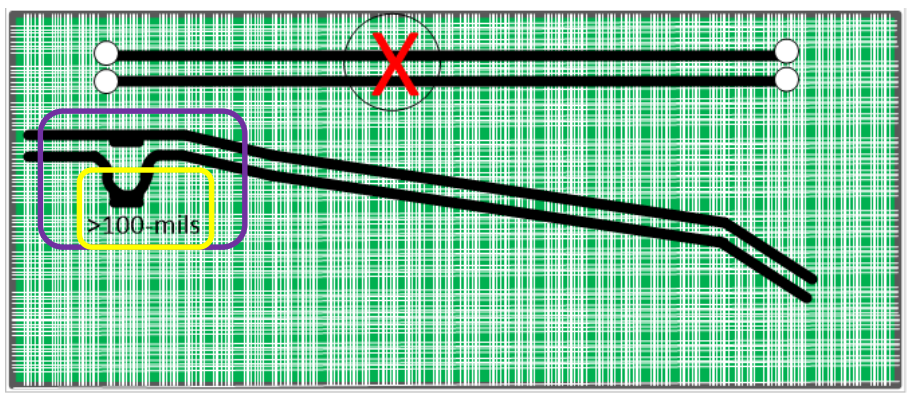
Trace length matching and trace length

- Avoid running long traces in parallel with grain of the fiber.
- Intra-pair trace should be matched to within 5-mils.
  - Trace mis-match compensation should be done at the point of mis-match.
- Trace width of any un-coupled section of a differential trace greater than 100-mils, should be increased to maintain target impedance



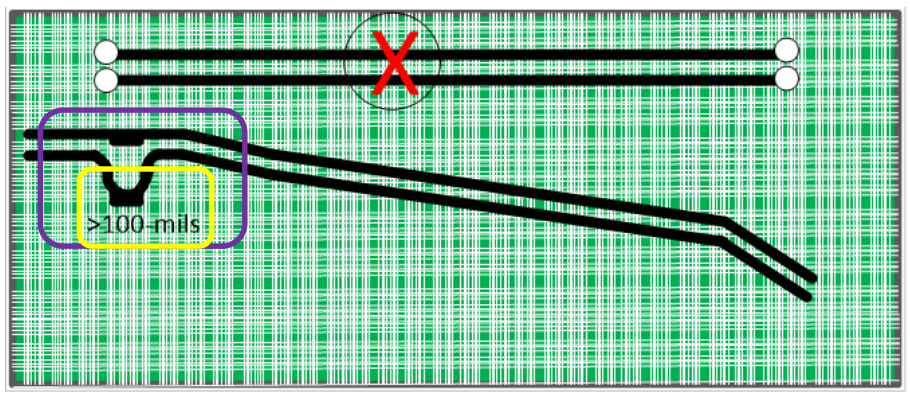
Trace length matching and trace length

- Avoid running long traces in parallel with grain of the fiber.
- Intra-pair trace should be matched to within 5-mils.
  - Trace mis-match compensation should be done at the point of mis-match.
- Trace width of any un-coupled section of a differential trace greater than 100-mils, should be increased to maintain target impedance
- Trace length to a connector is limited to 12 inches – this includes breakout area
  - Trace length from the edge-finger to add-in card chip set is limited to 4 inches
    - Board stack up and material can impact this length



Trace length matching and trace length

- Avoid running long traces in parallel with grain of the fiber.
- Intra-pair trace should be matched to within 5-mils.
  - Trace mis-match compensation should be done at the point of mis-match.
- Trace width of any un-coupled section of a differential trace greater than 100-mils, should be increased to maintain target impedance
- Trace length to a connector is limited to 12 inches – this includes breakout area
  - Trace length from the edge-finger to add-in card chip set is limited to 4 inches
  - Board stack up and material can impact this length



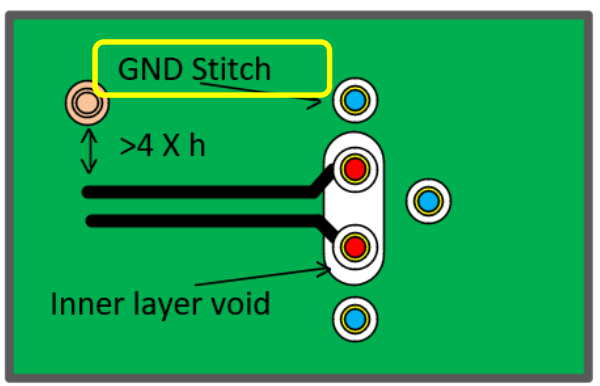
Via placement and guidelines

- Avoid vias as much as possible
  - Maximum of 6 via pairs are allowed on entire transmission line
  - In terms of loss, each via pair contributes to the loss budget per data rate 1dB at Gen4 and 0.5dB at Gen3
  - Via pad size should be 25 mils or less and a finished hole size of 14 mils or less

Via placement and guidelines

- Avoid vias as much as possible
  - Maximum of 6 via pairs are allowed on entire transmission line
  - In terms of loss, each via pair contributes to the loss budget per data rate 1dB at Gen4 and 0.5dB at Gen3
  - Via pad size should be 25 mils or less and a finished hole size of 14 mils or less

- Provide GND stitch to improve signal impedance transition
  - Location of via should be simulated. Else via should be placed at > via center to center spacing and < 100mils away from the via
  - Signal vias should have pads removed on unused internal layer
  - GND Stitch should be between the signal two reference planes

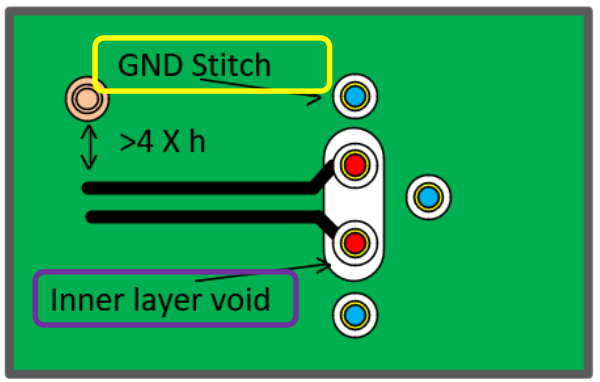


Via placement and guidelines

- Avoid vias as much as possible
  - Maximum of 6 via pairs are allowed on entire transmission line
  - In terms of loss, each via pair contributes to the loss budget per data rate 1dB at Gen4 and 0.5dB at Gen3
  - Via pad size should be 25 mils or less and a finished hole size of 14 mils or less

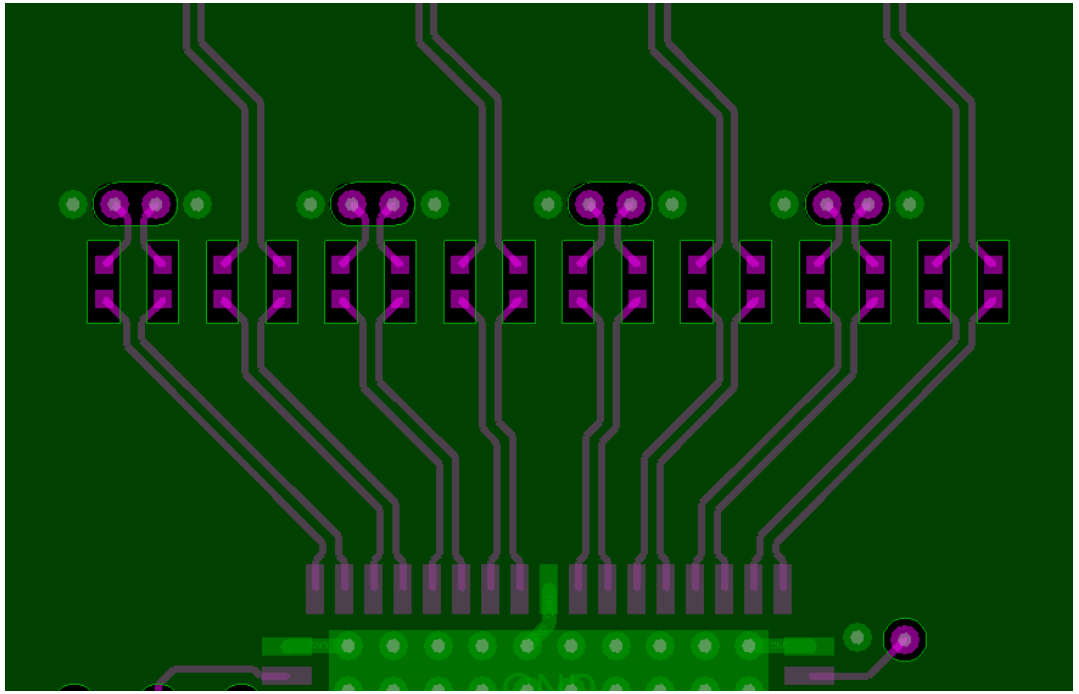
- Provide GND stitch to improve signal impedance transition
  - Location of via should be simulated. Else via should be placed at > via center to center spacing and < 100mils away from the via
  - Signal vias should have pads removed on unused internal layer
  - GND Stitch should be between the signal two reference planes

- Provide inner layer void – no GND or VCC – to reduce parasitic



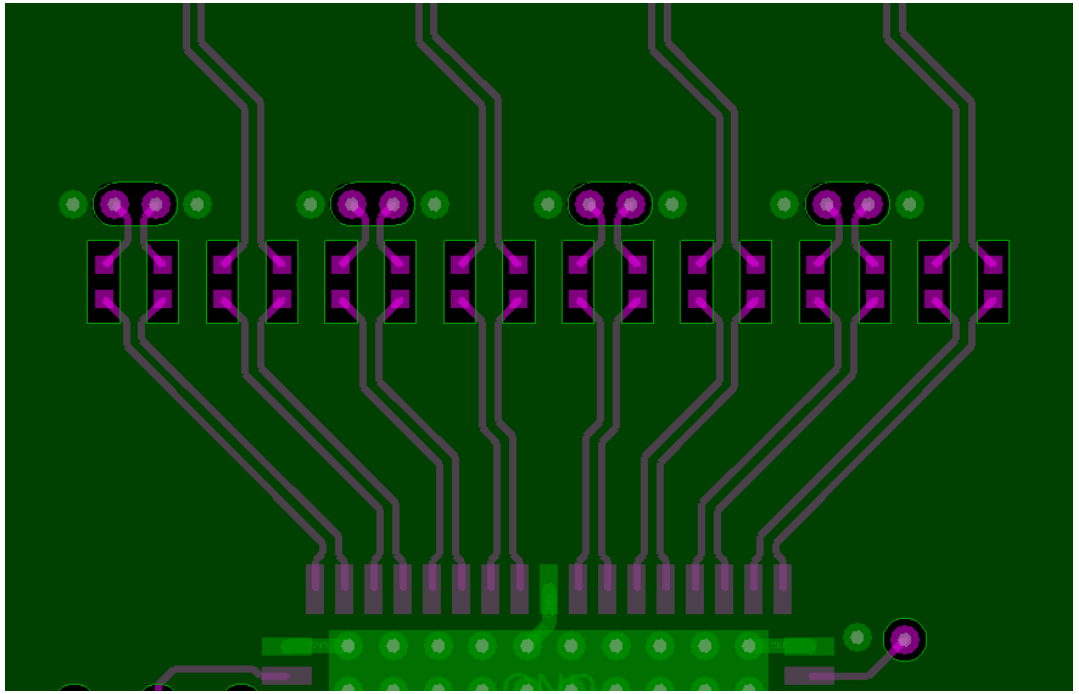
AC Coupling Cap void

- Void under AC coupling Caps pad:
  - Through as many layers as possible



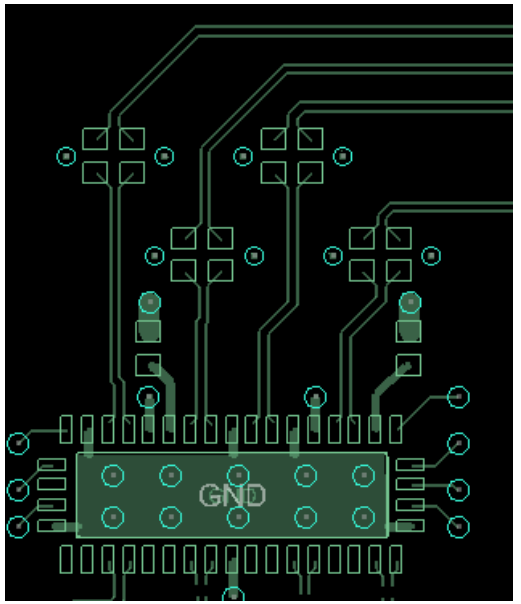
AC Coupling Cap void

- Void under AC coupling Caps pad:
  - Through as many layers as possible
- Via to differential trace should have at least 3 times differential trace spacing



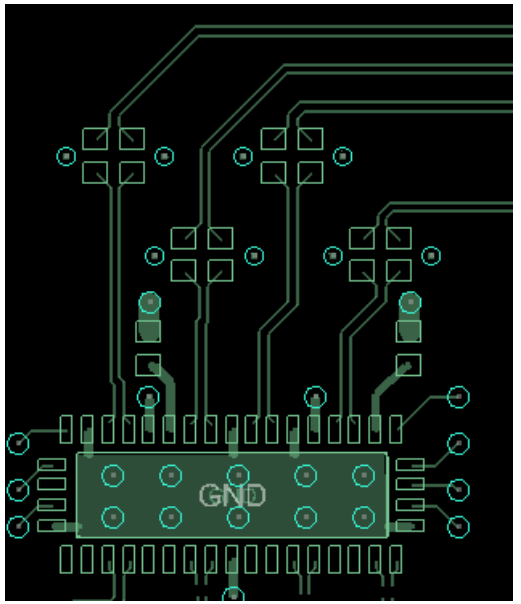
AC coupling capacitors placement

- Add-In Cards: Within 400mils of the CEM connectors



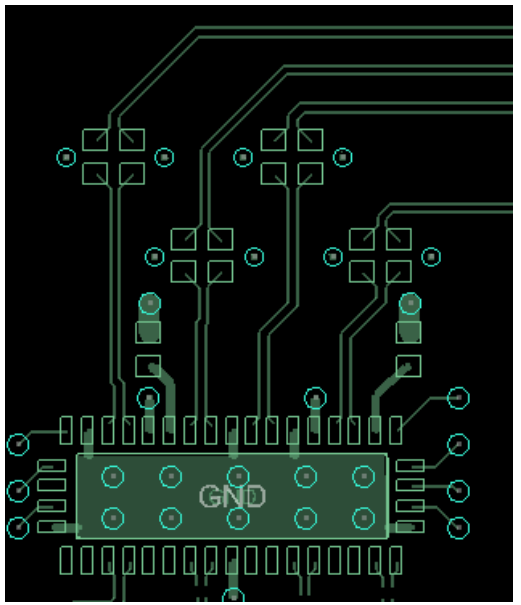
AC coupling capacitors placement

- Add-In Cards: Within 400mils of the CEM connectors
- Add-In Cards: Within 250mils of the finger edge



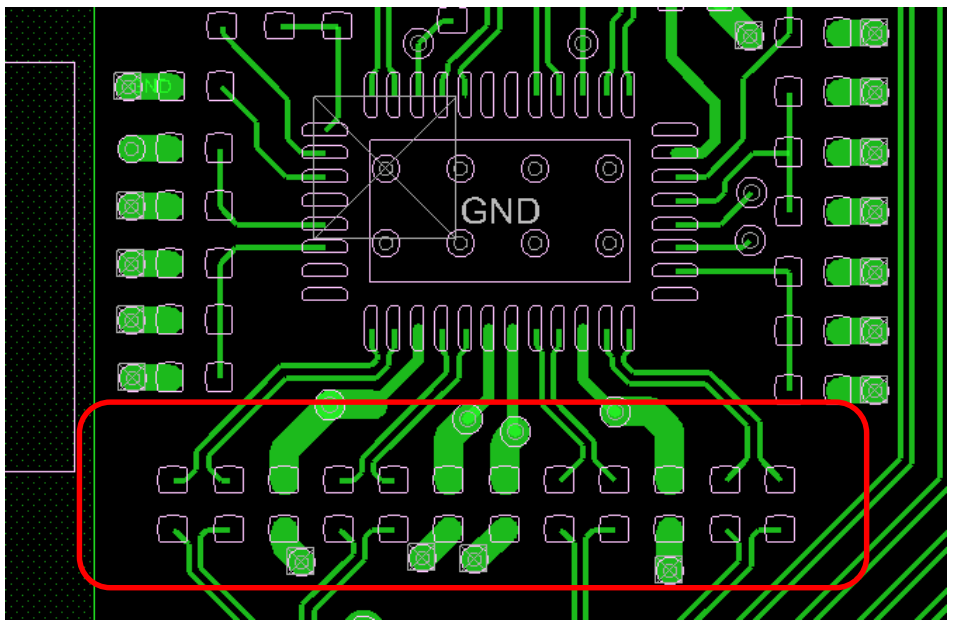
AC coupling capacitors placement

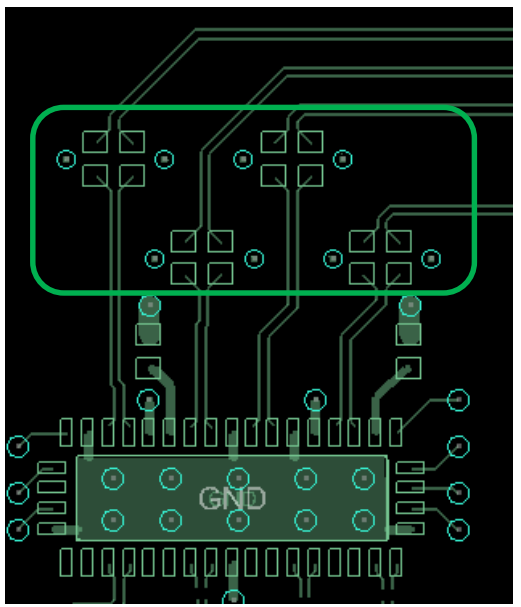
- Add-In Cards: Within 400mils of the CEM connectors
- Add-In Cards: Within 250mils of the finger edge
- Chip to chip connections, should be placed on the receiver side



AC coupling capacitors placement

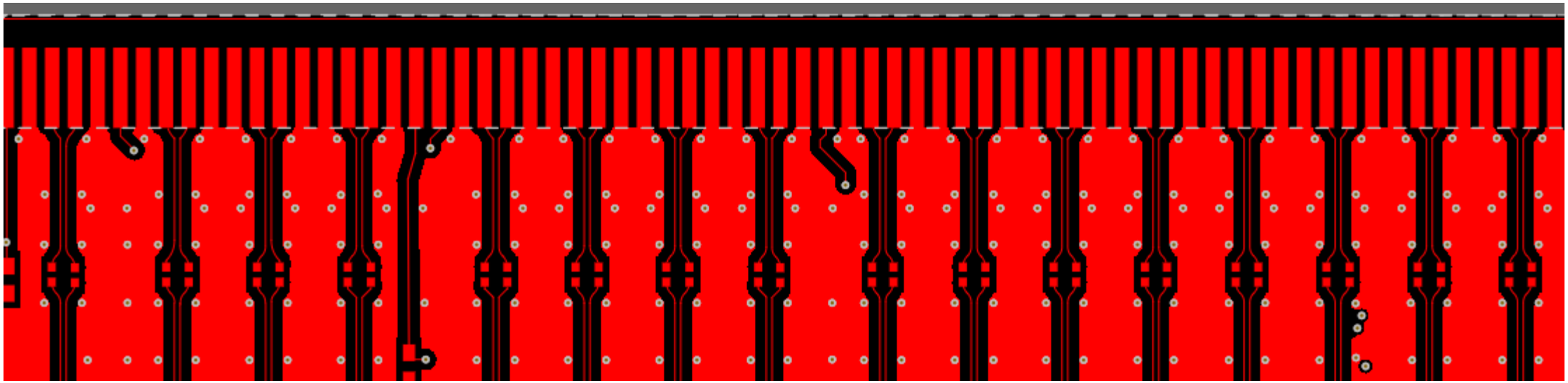
- Add-In Cards: Within 400mils of the CEM connectors
- Add-In Cards: Within 250mils of the finger edge
- Chip to chip connections, should be placed on the receiver side
- Should be staggered to reduce plane disruption as much as possible
  - Improves high speed signal isolation as well





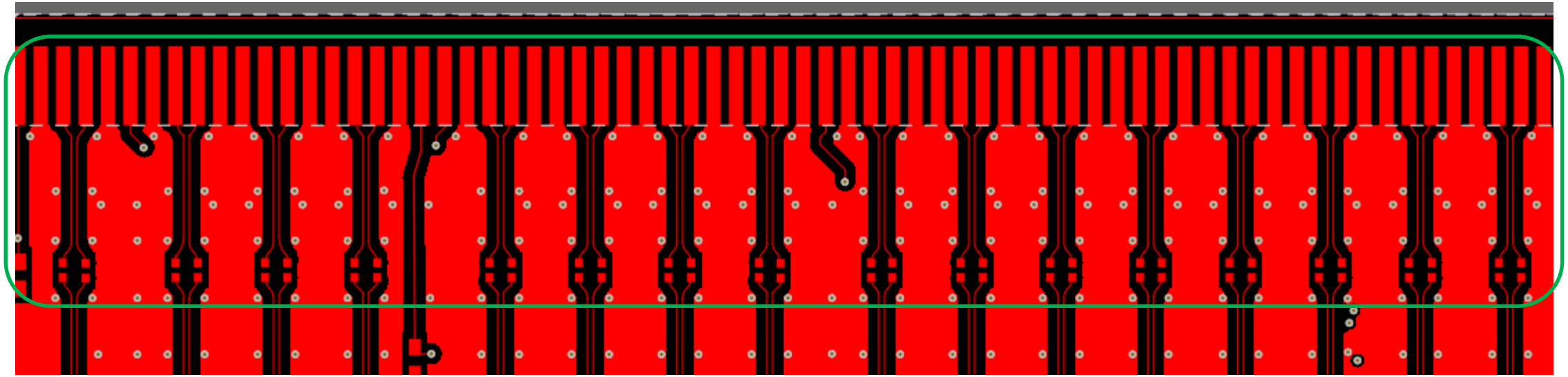
AC coupling capacitors placement

- Add-In Cards: Within 400mils of the CEM connectors



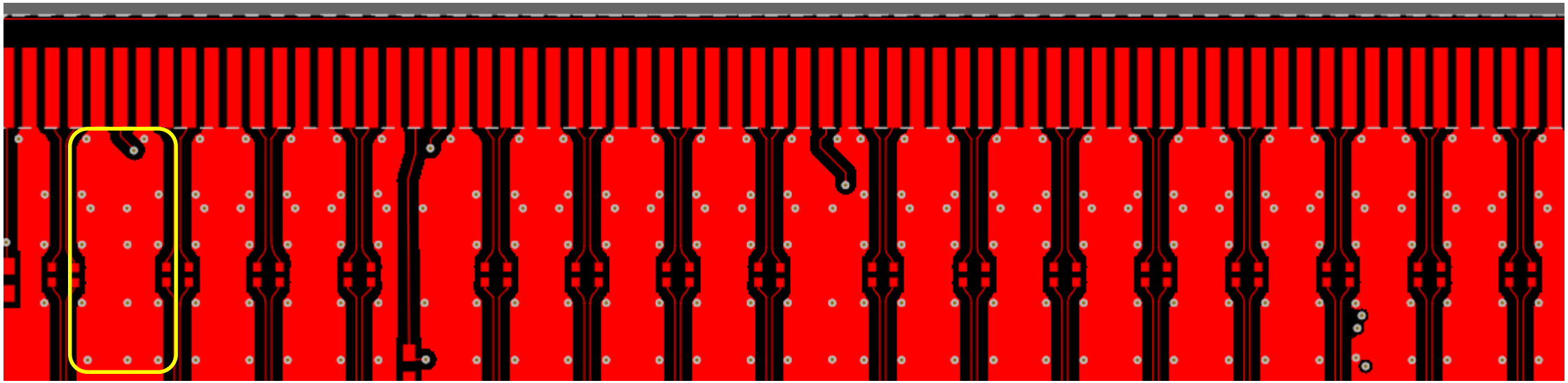
AC coupling capacitors placement

- Add-In Cards: Within 400mils of the CEM connectors
- Add-In Cards: Within 250mils of the finger edge



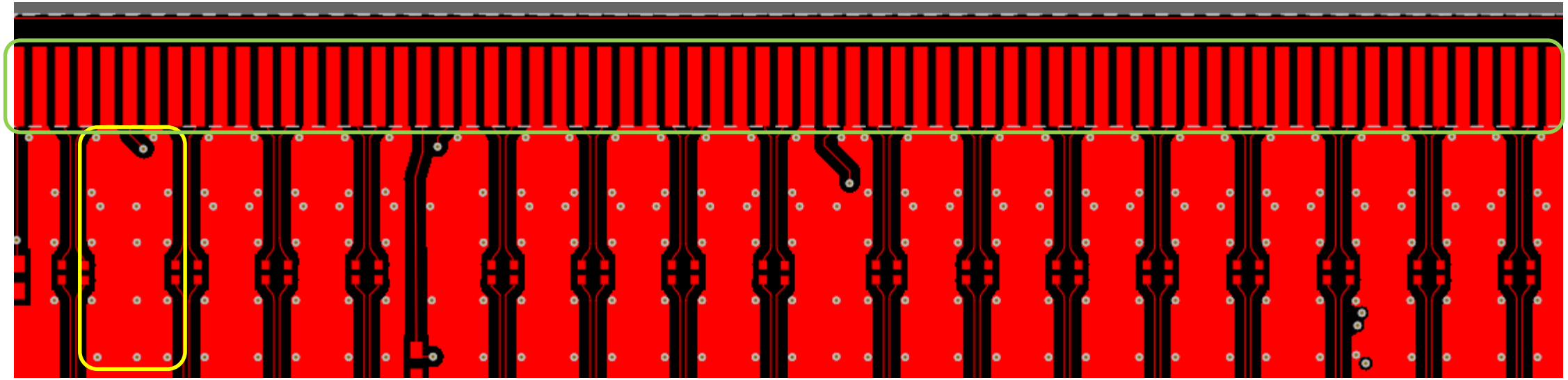
Cross talk mitigation

- GND Stitches along GND fill and vias



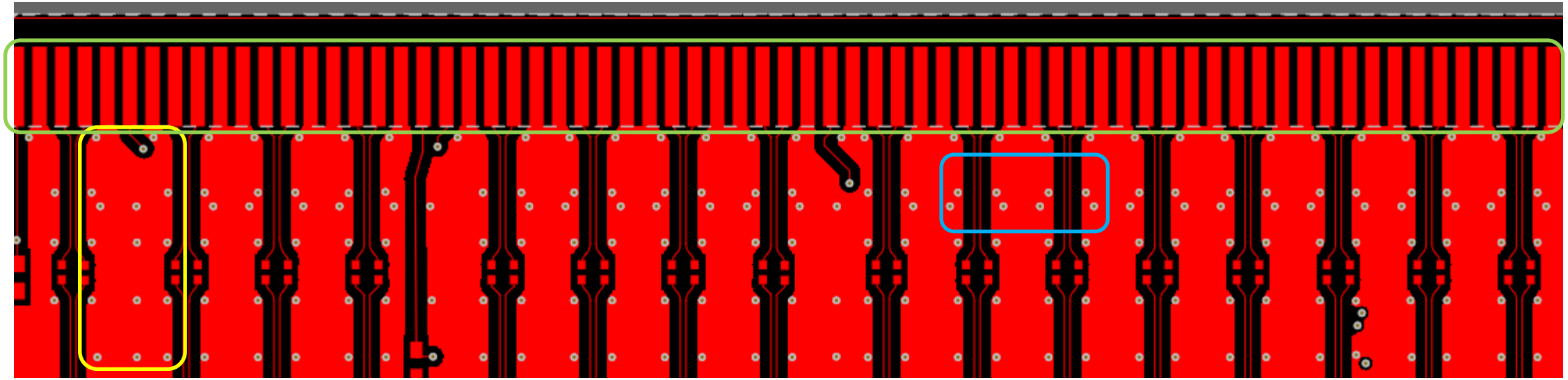
Cross talk mitigation

- GND Stitches along GND fill and vias
- Void under high speed pads – fingers and device



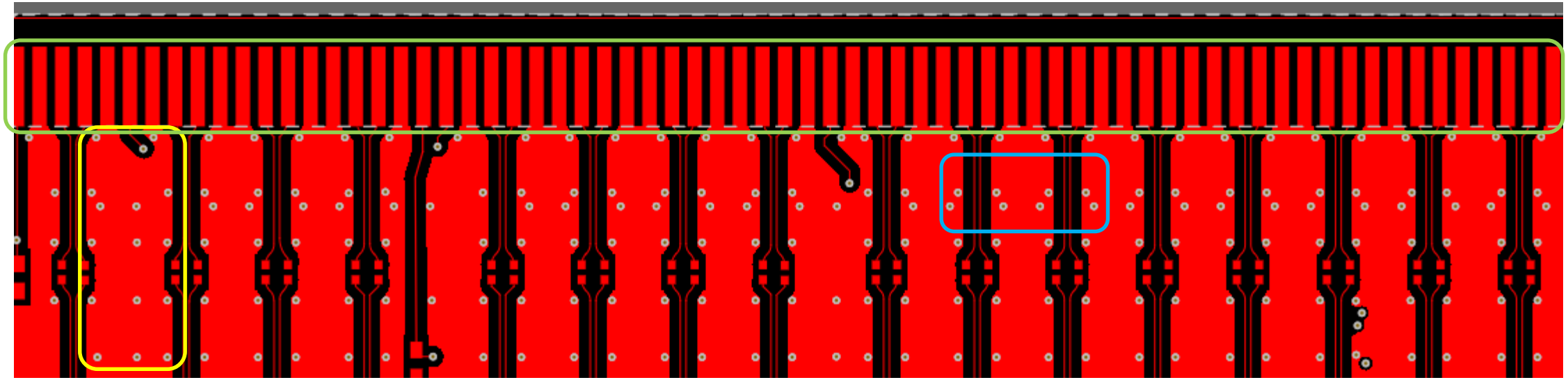
Cross talk mitigation

- GND Stitches along GND fill and vias
- Void under high speed pads – fingers and device
- Maintain differential trace impedance and distance from one another



Cross talk mitigation

- GND Stitches along GND fill and vias
- Void under high speed pads – fingers and device
- Maintain differential trace impedance and distance from one another



Thank you

- TI Precision Labs – PCIe Solving Signal Integrity Challenges