

# The Anatomy of a Differential Pair

Design Guidelines to Keep in Mind When Working with Differential Pairs

### PIN PITCH

- Choose whether to **Include** or **Ignore**
- If included, the etch gap or from etch gather point) is needed when calculating the max uncoupled length.

### MIN LINE WIDTH

- The trace width that should be used to route the differential pair nets the majority of the time
- The width you prefer your differential pairs to be routed

### PHASE TOLERANCE

- Allowable difference in length between the differential pair nets
- Phase tolerance is the timing (Flight Time) of the signal leaving the driver and getting to the receiver in phase (together)

### PRIMARY GAP

- The spacing, edge-to-edge, that should be used to route the differential pair nets the majority of the time
- The rule you prefer your differential pairs to follow
- This only applies to the two differential pair nets. Other net spacing to the differential pair nets is controlled by the Spacing Rule set Line to Line clearance

**Note:** The stackup of the board drives the line width and gap, so be sure to check in with your fabricators

### MAX UNCOUPLED LENGTH

- The trace width that should be used to route the differential pair nets the majority of the time.
- The width you prefer your differential pairs to be routed

### SEPARATION GAP TOLERANCE

*Coupled Tolerance (+)/(-)*

- Provides a coupling range based on the primary Separation Gap
- Summing Primary Separation Gap and Coupled Tolerance (+) provides the maximum coupled gap
- Subtracting Primary Separation Gap and Coupled Tolerance (-) provides the minimum coupled gap
- Values above or below these become an uncoupling event

### NECK WIDTH & NECK GAP

- Rules to be applied when the traces must “squeeze” down to be routed between pins/vias (for example, in BGA areas)
- Neck Gap is the new spacing, edge to edge, that should be used to route the differential pair
- Neck width is the new trace width that should be used to route the differential pairs

These differential pair design guidelines are just a few of the many items engineers must verify in their attempt to prevent sending a bad design to production. With the number of constraints and managed nets continuing to increase, keeping track of rules becomes an arduous task. Implementing a constraint-aware CAD environment allows design teams to embed their design intent directly into their CAD data at both the schematic and PCB stage. This ensures everyone stays in-sync throughout the design process. Simply put, a constraint-driven environment helps design teams reduce errors due to miscommunication and achieve production-ready designs faster.

For more information about a Constraint-Driven PCB Design Environment, visit:

[go.ema-eda.com/PCBConstraints](http://go.ema-eda.com/PCBConstraints)

ABOUT EMA DESIGN AUTOMATION, INC.

EMA Design Automation is a trailblazer in product development solutions offering a complete range of EDA tools, PLM integrations, services, training, and technical support. EMA is a Cadence® Channel Partner serving all of North America. EMA develops Ultra Librarian®, TimingDesigner®, CircuitSpace®, CIP™, EDABuilder®, and a host of custom solutions to enhance the OrCAD products, and all are distributed through a worldwide network of value-added resellers. EMA is a privately held corporation headquartered in Rochester, New York.

contact us: [info@ema-eda.com](mailto:info@ema-eda.com)