# HDI Via Design: Comprehensive Guide

## Overview

Vias are critical interconnects in HDI PCBs, connecting layers and enabling high-density routing. This guide covers via design, types, dimensions, and best practices for HDI applications.

## Via Types in HDI

### Through-Hole Vias (Traditional)

#### Characteristics

- Diameter: 0.3mm - 0.6mm typical
- Aspect Ratio: ≤10:1
- Connection: Through entire board
- Limitation: Consumes routing space

### Blind Microvias

#### Definition

Connect an outer layer to one or more inner layers without going through the entire board.

#### Characteristics

- Diameter: 0.075mm - 0.15mm (3-6 mil)
- Depth: One or more layers
- Aspect Ratio: ≤1:1 recommended
- Process: Laser-drilled or mechanically drilled

#### Benefits

- Space savings
- Improved signal integrity (shorter path)
- Better for high-frequency signals
- Reduced stub effects

## Buried Vias

### Definition

Connect inner layers without extending to outer layers.

### Characteristics

- **Diameter**: 0.15mm - 0.25mm
- **Depth**: Between inner layers
- **Process**: Created during lamination
- **Visibility**: Not visible from outer layers

## Stacked Microvias

### Definition

Multiple microvias aligned vertically, connecting multiple layers.

### Design Rules

1. **Alignment**: Precise registration required
2. **Fill**: Typically filled with conductive or non-conductive material
3. **Plating**: Continuous plating through stack
4. **Reliability**: Lower than staggered vias

## Staggered Microvias

### Definition

Offset microvias in adjacent layers, connected through traces.

#### Benefits

- Reliability: Better than stacked
- Manufacturing: Easier than stacked
- Cost: Similar to stacked

## Via Dimensions and Specifications

### Standard HDI Via Sizes

| Via Type | Drill Size | Finished Hole | Pad Diameter | Land | Aspect Ratio |
|---|---|---|---|---|---|
| Laser Microvia | 0.075mm | 0.10mm | 0.20mm | 0.25mm | ≤1:1 |
| Small Microvia | 0.10mm | 0.125mm | 0.25mm | 0.30mm | ≤0.8:1 |
| Standard Microvia | 0.15mm | 0.18mm | 0.30mm | 0.35mm | ≤0.8:1 |

## Via-in-Pad Design

### Benefits

#### Electrical

- Reduced Inductance: Shorter path
- Improved Signal Integrity: Minimized stubs
- Better High-Frequency Performance: Less impedance discontinuity

#### Mechanical

- Space Savings: No escape routing needed

- **Smaller Footprint**: Enables tighter component placement
- **Better for Fine Pitch**: Essential for <0.5mm pitch BGAs

## Design Process

### 1. Via Drilling

- Use laser drilling for small vias
- Ensure proper location accuracy
- Maintain minimum distance from pad edge

### 2. Plating

- Standard copper plating (25µm minimum)
- Ensure good coverage
- Verify plating quality

### 3. Fill Materials

#### Conductive Epoxy Fill

- Silver-filled epoxy
- Improves thermal conductivity
- Provides electrical path
- Used for thermal or ground vias

#### Non-Conductive Epoxy Fill

- Dielectric material
- Provides planar surface
- Does not conduct electricity
- Used for signal vias

# Via Current Capacity

## Approximation Formula

For external vias with 10°C rise:
$I = 0.8 \times d^{1.3}$

Where:

- I = Current in amps
- d = Via diameter in mils

## Via Current Capacity Table

| Finished Hole Size | 10°C Rise | 20°C Rise | 30°C Rise |
|---|---|---|---|
| 0.15mm (6 mil) | 0.7A | 1.0A | 1.3A |
| 0.20mm (8 mil) | 1.0A | 1.4A | 1.8A |
| 0.25mm (10 mil) | 1.4A | 1.9A | 2.4A |
| 0.30mm (12 mil) | 1.7A | 2.4A | 3.0A |

## Multiple Vias for High Current

- Use multiple vias in parallel
- Space evenly for current sharing
- Consider thermal relief

## Common Via Design Mistakes

| Mistake | Impact | Solution |
|---|---|---|
| Insufficient annular ring | Breakout, poor reliability | Follow IPC minimums |
| Poor aspect ratio | Plating voids, opens | Keep ≤1:1 for microvias |
| Inadequate clearance | Shorts, spacing violations | Check design rules |
| Via stubs in high-speed | Signal reflections | Use microvias or back-drill |
| No thermal relief | Soldering issues | Use thermal relief for plane connections |

References: IPC-2226, IPC-6016, IPC-6012, IPC-2152