

HDI Beginner Guide

HDI PCB: A Beginner's Guide to High Density Interconnect

!HDI PCB

What is HDI PCB?

High Density Interconnect (HDI) PCB represents a significant advancement in printed circuit board technology, designed to accommodate the increasing demand for compact, high-performance electronic devices. HDI boards feature:

- **Higher wiring density** per unit area compared to conventional PCBs
- **Smaller vias** (microvias) and finer lines/spaces
- **Multiple layers** with blind and buried vias
- **Advanced materials** for improved performance

Key Features of HDI PCBs

1. Microvia Technology

- **Laser-drilled microvias**: Typically 0.1mm or smaller
- **Via-in-pad**: Vias placed directly component pads
- **Stacked microvias**: Multiple microvias stacked vertically
- **Skip vias**: Vias that skip inner layers

2. Fine Pitch Design

- **Line/Space**: Down to 3/3 mil or less
- **Reduced layer count** while maintaining functionality
- **Smaller form factors** for portable devices

3. Advanced Layer Structures

- **Build-up layers**: Sequential lamination process
- **Any-layer HDI**: Microvias can connect any layer
- **Sequential lamination**: Multiple press cycles

Applications of HDI PCBs

HDI technology is essential for:

Consumer Electronics

- Smartphones and tablets
- Wearable devices
- Digital cameras
- Portable gaming devices

Computing

- Laptops and ultrabooks
- Mini PCs
- Graphics cards
- Memory modules

Communications

- 5G smartphones
- WiFi modules
- Bluetooth devices
- IoT sensors

Medical Devices

- Implantable devices
- Diagnostic equipment
- Monitoring devices
- Portable medical devices

Design Considerations for Beginners

1. Design Rules

Minimum Requirements:

Feature	Standard HDI	Advanced HDI
-----	-----	-----
Microvia Size	0.15mm	0.10mm
Laser Drill	0.10mm	0.075mm
Line/Space	4/4 mil	3/3 mil
Pad Size	0.30mm	0.25mm

2. Layer Stack-up Design

Best Practices:

- Use symmetrical stack-ups
- Balance copper distribution
- Place ground planes appropriately
- Consider thermal management

3. Via Selection

Types of Vias:

- **Through vias:** Connect all layers
- **Blind vias:** Connect outer layer to inner layers
- **Buried vias:** Connect inner layers only
- **Microvias:** Small laser-drilled vias

4. Material Selection

Key Considerations:

- **Dielectric constant:** Match your frequency requirements
- **Glass transition temperature (Tg):** High-Tg for reliability
- **Coefficient of thermal expansion (CTE):** Match component requirements
- **Surface finish:** ENIG, OSP, or immersion silver

Manufacturing Process

Step 1: Design

- Use HDI-capable CAD software
- Follow manufacturer's design rules
- Include proper documentation
- Perform DFM (Design for Manufacturing) checks

Step 2: Material Preparation

- Select appropriate laminate materials
- Prepare copper foils
- Clean and prep surfaces

Step 3: Laser Drilling

- **CO2 laser**: For dielectric layers
- **UV laser**: For copper layers
- **Via formation**: Precise diameter control

Step 4: Metallization

- **Desmear**: Remove drilling residue
- **Electroless copper**: Initial copper deposition
- **Electroplating**: Build up copper thickness
- **Via fill**: Optional conductive or non-conductive fill

Step 5: Patterning

- **Imaging**: Apply photoresist
- **Exposure**: Transfer circuit pattern
- **Development**: Remove unwanted resist
- **Etching**: Remove unwanted copper

Step 6: Lamination

- **Layer alignment**: Precision registration
- **Press cycle**: Temperature and pressure
- **Curing**: Complete polymerization
- **Multiple cycles**: For build-up layers

Common Challenges and Solutions

Challenge 1: Registration Accuracy

Solution:

- Use advanced registration systems
- Implement X-ray verification
- Maintain tight tolerances

Challenge 2: Via Reliability

Solution:

- Proper via cleaning
- Adequate copper plating
- Thermal cycling testing

Challenge 3: Signal Integrity

Solution:

- Controlled impedance design
- Proper reference planes
- Minimize via stubs

Challenge 4: Cost Management

Solution:

- Optimize layer count
- Use standard materials where possible
- Balance features vs. cost

Design Tips for Beginners

- **Start Simple:** Begin with 1+n+1 stack-up (1 build-up layer on each side)
- **Follow Guidelines:** Use manufacturer's design rules

- **Plan for Testing:** Include test points
- **Document Everything:** Keep detailed records
- **Prototype First:** Test before mass production

Testing and Quality Assurance

Electrical Tests

- **Continuity testing:** Verify all connections
- **Isolation testing:** Check for shorts
- **Impedance testing:** Verify controlled impedance
- **High-pot testing:** Dielectric strength

Visual Inspections

- **X-ray inspection:** Check via quality
- **Microsection analysis:** Verify plating quality
- **Dimensional verification:** Ensure proper sizing
- **Surface inspection:** Check for defects

Cost Considerations

Factors Affecting Cost

Factor	Impact	Notes
-----	-----	-----

Layer Count	Higher	More layers = more cost
Via Density	Higher	More microvias = higher cost
Material Choice	Variable	Advanced materials cost more
Tolerances	Higher	Tighter tolerances = higher cost
Volume	Lower	Higher volume = lower unit cost

Cost Optimization Tips

- Use standard layer stack-ups
- Minimize custom features
- Balance performance vs. cost
- Consider panel utilization

Future Trends

The HDI PCB market continues to evolve:

- **Smaller Features:** Continued miniaturization
- **Higher Density:** More components per area
- **Advanced Materials:** Low-loss, high-frequency materials
- **Integration:** Embedded components and passives
- **Automation:** Smart manufacturing and AI

Getting Started

For beginners looking to design HDI PCBs:

- **Learn the basics:** Understanding PCB fundamentals first

- **Choose software:** Select HDI-capable CAD tools
- **Study examples:** Analyze existing HDI designs
- **Partner with manufacturers:** Work with experienced fabricators
- **Start small:** Begin with simple HDI designs
- **Iterate:** Learn from each design cycle

Conclusion

HDI PCB technology enables the next generation of compact, high-performance electronic devices. While it presents unique challenges, understanding the fundamentals and following best practices will help you successfully design and manufacture HDI boards.

Remember: Start simple, learn continuously, and don't hesitate to seek expert advice when needed.

Sources:

- [HDI PCB Beginner Guide](#)
- [HDI PCB Design Comprehensive Guide](#)
- [Mastering HDI PCB Design](#)